I, Louise F. Merz, hereby submit this as part of the requirements for the degree of:

Masters of Science in:

Aerospace Engineering and Engineering Mechanics

It is entitled:

A Numerical Study of a Transonic Compressor Rotor at Large Tip Clearance.

Approved by:

Dr. Paul Orkwis
Dr. Shaaban Abdallah
Dr. Mark G. Turner
Dr. Lyle D. Dailey
A NUMERICAL STUDY OF A TRANSONIC
COMPRESSOR ROTOR AT LARGE TIP CLEARANCE

A thesis submitted to the
Division of Research and Advanced Studies
of the University of Cincinnati

in partial fulfillment of the
requirements for the degree of

MASTER OF SCIENCE

in the Department of Aerospace
Engineering and Engineering Mechanics
of the College of Engineering

2003

by

Louise Florence Merz
B.S., Virginia Commonwealth University, 2000

Committee Chair: Dr. Paul D. Orkwis
ABSTRACT

Experimental and numerical investigations have shown that flow in the tip region of high-speed, low aspect ratio, transonic compressors highly influences aerodynamic efficiency and operating range. Improved clearance flow predictions and understanding the flow physics are key to understanding and controlling off-design flow deterioration.

Numerical simulations of an aft swept, transonic compressor were found to prematurely predict stall at large tip clearance conditions. These numerical simulations utilize a basic and often used computational fluid dynamics model of the rotor with single block, structured H-type grid that incorporated rooftop tip treatment with steady calculations performed on isolated rotors. Two methods for predicting exit static pressure profiles were examined and determined to have minimal effect on performance predictions. Rotor exit static pressure profiles determined from stage calculations were chosen and various computational fluid dynamic (CFD) models were then compared to determine their influence on predictions. The models compared were: 1) steady isolated rotor, 2) unsteady isolated rotor, 3) unsteady stage. The $\kappa-\varepsilon$ and $\kappa-\omega$ turbulence models were also compared.

The near stall predictions were unsteady and exhibited complex large-scale oscillations. Unsteady rotor calculations provided more accurate one-dimensional data match and stage calculations provided more accurate exit radial profile data match but at a high computational cost. The largest difference in numerical results was observed for a difference in turbulence model mixing and blockage associated with the tip flow. The inability to obtain a converged steady-state solution at low flow rates indicates the significance of an unsteady phenomenon present in the flow field further supporting the need for unsteady flow simulations at off-design conditions. The current investigation has shown that unsteady isolated rotor simulations provide the most practical balance of accuracy in prediction and computational cost.
Although the stage simulations proved to be computationally expensive, the results exhibited unsteady behavior with frequencies other than blade passing that allude to the presence of a rotating type stall cell(s). Further investigation into the flow field that would include multiple passage simulations are recommended to capture the presence of rotating aperiodic flow phenomena and would prove useful to improving the understanding of tip flow during off-design conditions.
ACKNOWLEDGEMENTS

I would like to thank GE Aircraft engines for the opportunity and support provided through the USA program.

I would like to thank Dr. Lyle Dailey of the Engineering Tools and Development group at G.E. Aircraft Engines for his support and guidance throughout this Masters research program. I offer my immense gratitude for providing me the proper resources, encouragement and support during this research project.

I would also like to thank my thesis and academic advisor at the University of Cincinnati, Dr. Paul Orkwis. Dr. Orkwis provided the opportunity necessary to allow me to work at G.E. and provided me invaluable advice and guidance throughout this work.

In addition, I would like to thank all my colleagues at G.E., especially Allan Van deWall and Fu-Lin Tsung, for providing thoughtful discussions, ideas and insights throughout my research as well as encouragement that helped me finish the project.

I also thank my fellow graduate students, Bojan Lukovic and Greg Petrasko, who not only provided academic assistance, but amusement for the past two years.
CONTENTS

Abstract i
Acknowledgements v
Table of Contents vi
List of Figures viii
List of Tables xi
Nomenclature xii

1 Introduction 1

1.1 Purpose for Off-Design Investigation 2

1.2 Axial Compressors and Fans 3

1.3 Surge / Stall 3

1.3.1 Stall Flow Physics 4

1.3.2 Stall Margin 5

1.3.3 Stall Recovery 6

1.3.4 Stall Correlations 7

1.3.5 Stability Enhancement 10

1.4 Tip Clearance Flow 12

1.5 Tip Clearance Modeling 16

1.5.1 Turbulence Modeling 17

1.5.2 Viscous Effects 18

2 Background on Wennerstrom Design 20
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3 Motivation and Approach Background</td>
<td>24</td>
</tr>
<tr>
<td>4 Numerical Method</td>
<td></td>
</tr>
<tr>
<td>4.1 Algorithm</td>
<td>29</td>
</tr>
<tr>
<td>4.2 Boundary Conditions</td>
<td>30</td>
</tr>
<tr>
<td>4.3 Computational Grids</td>
<td>31</td>
</tr>
<tr>
<td>5 Results and Discussion</td>
<td>33</td>
</tr>
<tr>
<td>5.1 One-Dimensional Performance</td>
<td>33</td>
</tr>
<tr>
<td>5.2 Turbulence Model</td>
<td>47</td>
</tr>
<tr>
<td>5.3 Isolated Rotor – Steady/Unsteady</td>
<td>59</td>
</tr>
<tr>
<td>5.4 Stage/Isolated Rotor</td>
<td>69</td>
</tr>
<tr>
<td>6 Conclusions and Recommendations</td>
<td>81</td>
</tr>
<tr>
<td>References</td>
<td>83</td>
</tr>
</tbody>
</table>
# List of Figures

1.1 Schematic of Solidity and Aspect Ratio  
1.2 Schematic of a Stall Cell  
1.3 Schematic of Stall Margin  
1.4 Effect of Hysteresis  
1.5 Schematic of Stage Geometry  
1.6 Schematic of Tip Clearance  
1.7 Effect of Spacing Between Blade Rows on Stalling Static Pressure Rise Coefficient [2]  
1.8 Effect of Tip Clearance on Stalling Static Pressure Rise Coefficient [2]  
1.9 Schematic of Tip Clearance Vortex and Shock/Vortex Interaction  
1.10 Schematic of Tip Clearance Vortex and Shock/Vortex Interaction at Reduced Mass Flow  
3.1 Predicted Static Pressure Profiles at Rotor Exit  
3.2 Calculated Total Pressure Profiles at Rotor Exit.  
4.1 Blade-to-Blade View of Stage Computational Grid  
4.2 Rotor Leading and Trailing Edges  
4.3 Computational Grid – for Wedge Tip Treatment  
5.1 One-Dimensional Performance – Pressure Ratio Map  
5.2 One-Dimensional Performance – Temperature Ratio Map  
5.3 One-Dimensional Performance – Efficiency Map  
5.4 Points Used for Comparison of Total Pressure and Total Temperature profiles Downstream of the Rotor at Peak Efficiency
5.5 Comparison of Total Pressure and Total Temperature Profiles Downstream of the Rotor at Peak Efficiency

5.6 Steady, Isolated Rotor Inlet Corrected Mass Flow Histories for Three Flow Conditions

5.7 Steady and Unsteady $\kappa-\varepsilon$ Isolated Rotor Inlet Corrected Mass Flow Histories For Three Flow Conditions

5.8 Unsteady Stage Inlet Corrected Mass Flow Histories for Two Flow Conditions

5.9 Pressure Ratio Map Indicating Turbulence Model Comparison Operating Points

5.10 Relative Mach Contours at Blade Tip for Steady, Isolated Rotor Simulations


5.12 Tip Vortex Streamlines for Steady, Isolated Rotor Simulations

5.13 Vorticity Contours for Steady, Isolated Rotor Simulations

5.14 Vorticity Contours and Velocity Vectors Taken Perpendicular to Tip Flow Vortex for Steady, $\kappa-\omega$ Isolated Rotor Simulation at 15.1 kg/s Exit Corrected Mass Flow.

5.15 Turbulent Viscosity Contours at Blade Tip for Steady, Isolated Rotor Simulations

5.16 Turbulent Kinetic Energy Contours at Blade Tip for Steady, Isolated Rotor Simulations

5.17 Pressure Ratio Map Indicating Steady/Unsteady Isolated Rotor Comparison Operating Points

5.18 Relative Mach Contours at Blade Tip for Isolated Rotor, Steady and Unsteady Simulations

5.19 Contours of Negative Axial Velocity at Blade Tip for Isolated Rotor, Steady and Unsteady Simulations.

5.20 Mass Flow History for Steady and Unsteady, Isolated Rotor Simulations at Exit Corrected Flow of 15.7 kg/s.
5.21 Instantaneous Contours of Negative Axial Velocity at Blade Tip for Isolated Rotor, Steady and Unsteady Simulations at Exit Corrected Flow of 15.7 kg/s

5.22 Instantaneous Contours of Relative Flow Angle Measured from Axial Direction at Blade Tip for Isolated Rotor, Steady and Unsteady Simulations at Exit Corrected Flow of 15.7 kg/s.

5.23 Entropy contours at Blade Tip for Isolated Rotor, Steady and Unsteady Simulations

5.24 Pressure Ratio Map Indicating Stage Comparison Operating Points

5.25 Measured and Unsteady Stage Simulation Casing Static Pressure Contours Near Peak Efficiency

5.26 Relative Mach Number Contours at Blade Tip for Low and high Flow Unsteady Stage Simulations

5.27 Inle Corrected Mass Flow History

5.28 Rotor Exit Total Pressure Profiles for the Low Flow Unsteady Stage Calculation for the Full Range of Exit Corrected Flow Values

5.29 Comparison of Total Pressure and Total Temperature Profiles Downstream of the Stator at Peak Efficiency for the Low Flow Unsteady Stage Calculation for the Full Range of Exit Corrected Flow Values

5.30 Relative Mach Number and Negative Axial Velocity Variations at the Blade Tip for Low Flow Unsteady Stage Simulation
List of Tables

3.1 Matrix of Simulations Performed 28
Chapter 1

INTRODUCTION

Propulsion systems have improved significantly over the last 50 years in part through the use of computers, computational design tools, aerodynamic and mechanical design advances, design techniques, materials and manufacturing. Designers have used these tools and advancements in an effort to reduce weight, improve ruggedness and improve performance in machine design. One trend is to increase the amount of work extracted per stage without deteriorating performance, which has resulted in overall pressure ratios across a single compressor spool doubling since 1955 [1]. Recent compressor and fan design trends are toward greater speeds, larger pressure rise per stage, and higher aerodynamic loading through the use of improved blading shapes, blade configurations and lower aspect ratios (Figure 1.1). This drive to larger pressure rise per stage has also been paralleled with increased efficiency. However, as compressors have improved at the design point, good off-design performance must also be maintained. The rotor investigated in the current study is a low-aspect ratio, transonic compressor rotor and is inclusive of the current trends. The following subsections of the introduction will examine the purpose for off-design investigations, a presentation of compressor and fan design intent and a discussion of stall and surge. This will be followed by a discussion of tip clearance flow, its impact on off-design performance and typical tip clearance flow modeling techniques and concerns.
1.1 Purpose for Off-Design Investigation

Compressors and fans may be designed for a target operating condition but must operate safely and reliably under off-design conditions. A commercial aircraft engine is designed for cruise conditions, but must also perform during take-off and landing as well as when condition variations are encountered in flight. Compressors and fans often encounter inlet distortion, throttle and/or speed changes, geometry changes, material changes from temperature or erosion and foreign object ingestion during operation. The allowable range of off-design conditions is often reduced with advanced designs making them more likely to experience detrimental conditions like rotating stall (circumferentially rotating passages with reduced mass flow) and surge (axially varying mass flow). A gas turbine engine will experience performance and durability problems if it is unable to avoid rotating stall and surge over its operating range. The circumferentially varying nature of rotating stall and the initiation of surge can place large transverse loads on the rotor and casing, resulting in blade rubbing and other damage. Losses can be reduced and causes of stall may be avoided by analyzing and

![Figure 1.1: Schematic of Solidity and Aspect Ratio](image)

Solidity = $\frac{c}{S}$

Aspect Ratio = $\frac{h}{c}$
understanding the physics at this point in engine operation. To that end, it is vitally important that computational simulation tools provide accurate results for off-design conditions. Before proceeding with a discussion of compressor off-design performance, it is important to review the intent of compressor and fan design. The next section presents the basic physical processes within a compressor and will serve as a preface to analyzing off-design conditions.

1.2 Axial Compressors and Fans

The purpose of a compressor or fan is to increase the pressure of the working fluid. Axial compressors generally accomplish this with a series of alternate rotating (rotors or blades) and stationary (stators or vanes) airfoil rows. In these configurations, the compression process is a two part process involving a rotor/stator combination, or stage. The rotor increases the total pressure of the fluid by adding kinetic energy (increasing tangential momentum), whereas both the rotor and stator turn the flow and increase the static pressure of the fluid through aerodynamic diffusion. Since the blade passage of a compressor acts as a diffuser, the boundary layers on all walls are subject to an adverse pressure gradient, i.e., an axial pressure increase. The performance of these machines is limited by the ability to control the boundary layer during diffusion and flow turning in the presence of this adverse pressure gradient. If the pressure gradient is not controlled, separation or stall will occur.

1.3 Surge / Stall

As mentioned previously, stall and surge are very serious problems that can be disastrous to performance, and recovery from these conditions can be difficult. At a constant rotational speed, an increase in the compressor pressure ratio is obtained by restricting the flow through (i.e., throttling) the compressor. As a compressor is throttled beyond its peak pressure rise, fundamental changes in the flow pattern take place and the compressor begins to stall or surge [2]. Stall may refer to a fully developed rotating stall, a transient condition initiating either
a surge or a large deviation in pressure loss or reduced flow turning [2]. While in the case of rotating stall a circumferentially non-uniform flow pattern rotates around the annulus; surge refers to an essentially one-dimensional overall mass flow variation in the annulus that may transition alternately into and out of stall.

1.3.1 Stall Flow Physics

As the compressor is throttled up, the mass flow reaches a point at which it is too low for the geometry, causing the incoming flow angle to deviate from design intent and resulting in flow separation on the blade surface. To compensate for the reduced mass flow, the stream becomes unequally divided amongst the blade passages resulting in a circumferential redistribution of the flow. Around the circumference, non-uniform regions of stagnant flow rotate around the annulus without moving axially [2]. These regions of stagnant flow usually originate near the endwall. More than one stall region, or cell, may be present and could extend to part

![Figure 1.2: Schematic of a Stall Cell](image)
or full span. When a cell is present it creates a decreased mass flow, which is comparable to a reduced flow area, or blockage. This blocked area reduces the blade incidence on one side and increases it on the other (see Figure 1.2) resulting in the cell rotating (usually) in the direction of increased incidence, i.e. backward relative to the rotor, at a fraction of the rotor speed [1,2]. The stagnant flow in one passage induces stall in the next passage by increasing its incidence, which in turn reduces the incidence in the original passage (unstalling it), thus causing the rotation of the stall cell.

1.3.2 Stall Margin

The performance of a compressor can be visualized using a performance map depicting pressure ratio (or other pressure function such as pressure coefficient) versus the airflow rate (Figure 1.3). As a compressor is throttled up a constant speed line (at a constant RPM), the pressure rise eventually reaches a level too high for the design speed of the compressor and the compressor will become unstable (stall or surge). The line defining the location of stall for different rotational speeds is called the stall line and represents the limiting boundary of stable operation.
compressor operation. The difference between the stall and operating lines (usually peak efficiency condition) is referred to as the stall (or surge) margin and is an indication of the stable operating range of the machine. It provides a quantifiable safety margin between the expected range of operation and stall and surge. Cumpsty [2] provides Equations 1.1 and 1.2 to define

\[ \text{Stall Margin} = \left[ F_{\text{out, operating}} - F_{\text{out, stall}} \right] / F_{\text{out, operating}} \quad (1.1) \]

\[ F_{\text{out}} = F_{\text{corrected}} = \dot{m} \sqrt{c_p T_0} / A p_0 \quad (1.2) \]

the stall margin and corrected mass flow (or flow function, \( F \)). A mass flow corrected to standard day conditions (corrected mass flow) is used to relate the airflow to the area of the throttle where choking occurs. A line of constant exit corrected flow is equivalent to a constant throttle setting, such as the operating line in Figure 1.3, and is representative of a point of operation. It should be noted here that the term stall margin is inclusive of both surge and stall phenomena. Stall margin can range from 15 to 20 percent in commercial and 20 to 30 percent in military engines [1]. By observing Figure 1.3, it can easily be seen that lowering the operating line or raising the stall line can increase the stall margin. Lowering the operating line is undesirable because doing so results in a lower pressure rise [2]. The more attractive choice for increasing the stall margin is, therefore, to raise the stall line as well as to reduce any increase in the working line during transients [2]. To improve the stall range of modern compressors an improved understanding of the flow during off-design conditions is needed; hence, accurate off-design prediction methods must be developed.

1.3.3 Stall Recovery

Surge and rotating stall phenomena are among the most serious problems in turbomachinery and the ability of the engine to recover is critical. Recovering from stall or surge
is not always straightforward. In returning a multistage compressor to its pre-stalled condition it may encounter a pressure ratio-mass flow characteristic (path taken or behavior during throttle) that is not unique. This effect is called hysteresis (Figure 1.4) and requires that the mass flow must be increased, by opening the throttle area, beyond its pre-stalled flow condition for the compressor to shift from the stalled to the unstalled characteristic. The deviation is a result of the process required for the separated flows and instabilities associated with stall to reattach and clear. A large hysteresis can hinder recovery from rotating stall or surge. Non-recoverable stall has been encountered and the difficult process of shutting down and re-igniting the engine has been necessary for recovery. The difficulties encountered in recovering from stall provide additional evidence of the need to avoid these conditions; therefore, accurate prediction of stall inception is critical.

1.3.4 Stall Correlations

The benefits to predicting and recovering from stall have been presented. Many variables contribute to the conditions that can lead to stall and it is useful to correlate these
factors in an effort to assess the likelihood of stall at a given flow rate. The conditions under which stall occurs are dependent on the boundary layers within the passage (whether they are separated or not), the presence and interaction of shocks, Reynolds number and blading. Koch [3] developed a non-dimensional parameter (Equation 1.3) to correlate blade geometry and the

\[
\frac{L}{g_2} = \frac{\text{Diffusion Length}}{\text{Exit Passage Width}} = \frac{C}{360} \frac{2\pi}{\theta} \frac{\sin \frac{\theta}{2}}{\frac{A_2}{A_1} Sh_1 \cos \beta_1 h_2}
\]

pressure rise capability of low-speed axial flow compressor stages by relating the performance of passages to that of a straight diffuser, in which the pressure rise capability is a function of its two-dimensional length to inlet width ratio. The corresponding length in a compressor must account for the curvature of the path and Koch used the meanline length of the circular-arc airfoil \(L\) that is calculated using the chord length \(C\) and the blade curvature \(\theta\). For a compressor, a characteristic dimension of the flow’s cross-sectional area (inlet width for a diffuser) was defined using the cascade trailing edge staggered spacing \(g_2\) which is calculated using the fact that the area \(A\) is equal to the staggered blade spacing \(g\) by the blade height \(h\) and the staggered spacing is equal to the tangential blade spacing \(S\) by the cosine of the relative flow angle \(\beta\) (see Figure 1.5). An exit dimension was used because it remains nearly constant over the operating range. This correlation also accounts for tip clearance (Figure 1.6), axial gap spacing between rotor and stator, Reynolds number and velocity diagram effects by adjusting the data to average quantities. The correlation was then compared to high-speed compressors and was found to be consistent for low and high speed machines. Koch’s correlation provides a reasonable estimate for the maximum pressure rise capability of a multistage compressor; however, Cumpsty [2] points out that many compressors show a compressor rise below (very few above) the correlation and that 80 percent of the value
predicted by Koch would be a more realistic assumption and is appropriate for preliminary design studies. In addition to providing a method for estimating pressure rise limitations, Koch showed the relationship each of these variables play in pressure rise capability of compressors. Ways of improving pressure rise capability through blade geometry are discussed next.

![Figure 1.5. Schematic of Stage Geometry](image1)

![Figure 1.6: Schematic of Tip Clearance](image2)
1.3.5  Stability Enhancement

Blading with higher peak pressure rise capability can be achieved through increased
solidity, reduced aspect ratio and reduced tip clearance [1,2]. Solidity (Figure 1.1) can be
increased by increasing the number of airfoils in a blade row or increasing the chord, which also
reduces the aspect ratio. These measures cause the machine to distribute the diffusion over a
longer area, effectively reducing the aerodynamic loading at a given point. A smaller load
reduces the chance of flow separation and stall through a reduced pressure gradient on both
the endwall and airfoil boundary layers [1]. A drawback to increased solidity and reduced
aspect ratio is an increase in friction drag and weight that act to reduce efficiency, thus a
balancing point must be reached. The effects of axial spacing and tip clearance level on stalling
static pressure rise used by Koch [3] are presented in Figures 1.7 and 1.8. Figure 1.7 shows
how the effect of axial spacing becomes less significant as the axial gap is enlarged while
Figure 1.8 shows how the effect of tip clearance continues to be significant throughout the levels
included indicating that tip clearance spacing remains a significant feature at all levels. In
addition to the effects of blade geometry, the number of blade rows or stages must be
considered because various stages (or individual blade rows) may be operating under stalled or
unstalled conditions. Compressor characteristics at off-design conditions and, specifically, the
point of instability are dependent upon the volume dynamics and flow characteristics of the
system [2,4,5] and so the number of rows and the spacing between rows of airfoils must be
included in performance prediction.
Figure 1.7: Effect of spacing between blade rows on stalling static pressure rise coefficient [2].

Figure 1.8: Effect of tip clearance on stalling static pressure rise coefficient [2].
To summarize the above, improved understanding of off-design performance will assist in creating designs with increased operating range without encountering stall, while causing minimal detriment to efficiency. What is known currently about off-design conditions is that the flow in the tip region and near the casing of high-speed, low aspect ratio transonic compressors and fans influences greatly the aerodynamic efficiency and range of operation and is key to understanding off-design flow deterioration. Studies show that current high-speed, low-aspect ratio compressor performance and flow characteristics are highly sensitive to the size of the tip clearance with reductions in performance as a consequence of increased tip clearance levels [3,6,7,8,9]. L. H. Smith [10] observed that a one percent increase in the clearance-to-chord ratio resulted in a five percent loss in peak pressure rise for subsonic test data. This finding was later found to be valid for a high-speed rotor through numerical experiments by Adamczyk [8]. This is an important observation in that the tip clearance will vary during operation due to thermal and load effects caused by flexing of the rotor and/or casing in addition to mechanical wear changes during a typical life cycle, resulting in a wide range of conditions experienced that are different from the design intent. The nature of the tip clearance flow is examined next, followed by methods for modeling the flow in this region.

1.4 Tip Clearance Flow

Experimental and numerical investigations have shown that the flow in the tip region and near the casing of high-speed, low aspect ratio, transonic compressors and fans highly influences aerodynamic efficiency and range of operation. Improved tip clearance flow predictions and understanding of the flow physics are key to understanding and controlling off-design flow deterioration.

Current knowledge of tip flow starts by considering spillage from the pressure side of the passage to the suction side of the passage caused by the pressure gradient. As the lower axial momentum tip leakage meets the main passage flow on the suction side of the blade, the axial
mass flow toward the tip is decreased with the deterred flow increasing the axial flow rate toward the center of the passage as opposed to increasing the pressure side flow. This reduced mass flow is comparable to a decrease in the flow area, or blockage, and results in less work done by the rotor. The difference in magnitude and direction of the tip leakage and the core passage flows creates a vortex sheet, which rolls up into the leakage vortex as it progresses downstream (Figure 1.9). The extent to which the tip flow influences the overall flow structure within the compressor is dependent on the size of the tip clearance, passage shock system, casing boundary layer, blade loading and the presence and behavior of the leakage vortex. An increase in tip clearance impacts the structure of the passage flow in that it allows for a larger mass flow to pass over the tip and creates a stronger tip vortex; this flow can assist in the creation of loss-associated secondary flows within the blade passage. As it progresses downstream, the leakage vortex grows from near the suction side leading edge and creates a region of high loss downstream of the leading edge. When a shock is present, as in the case of a transonic rotor, it interacts with the casing boundary layer and the leakage vortex increasing the possibility of casing boundary layer separation on the suction side of the blade tip (Figures 1.9 and 1.10). The structure of the tip flow changes as the mass flow is reduced with the tip vortex and shock pushing forward in the passage (Figure 1.10). Losses in this region are associated with boundary layer separations that exist at the blade tip (both within the gap and on the suction surface) as well as on the casing that result from flow aerodynamics and possible shock-boundary layer interaction. These observations of the tip clearance flow are indications of the flow complexity in this region that add to the difficulty of prediction.
Figure 1.9. Schematic of tip clearance vortex and shock/vortex interaction.

Figure 1.10. Schematic of tip clearance vortex and shock/vortex interaction at a reduced mass flow.
In numerical investigations, Hofmann and Ballmann [11] as well as Jennions and Turner [12] observed the passage shock interacting with the leakage vortex. Hofmann and Ballman show this interaction causing the total pressure in the core of the vortex to decrease and the vortex to spread, thus increasing blockage in the passage. As mass flow is reduced, the low energy (low axial momentum) fluid near the casing associated with the tip clearance flow moves upstream (negative axial velocity) and sometimes forward of the leading edge. Jennions and Turner [12] show changes in the flow pattern as the flow is reduced from peak efficiency toward stall. The changes in the flow include the shock detaching and the shock and vortex pushing upstream and intersecting the passage inlet at numerical stall conditions. Spilt tip leakage flow forward of the leading edge has been associated with the onset of stall [8], although the cause/effect relationship of the forward spilling flow and the instability has yet to be understood. Many methods have been implemented in an effort to improve or control the flow in the tip region, resulting from the large influence tip flow has shown on performance. Trenching, where the blade extends into a cutout in the casing, has been used to improve efficiency, although improvements are not seen in all cases, indicating that the very complex local flow physics are not completely understood. Another approach is to implement a tip or casing treatment such as grooves or slots in the tip to direct the flow. The presence of grooves has provided improvement in stall margin at a cost to efficiency [13]. Lee and Greitzer [13] successfully used endwall suction and blowing to remove blocked flow and improve overall stall margin. Although the success of tip treatment approaches is not universal or completely understood, results have implied that the causes of stall lie in the rotor tip region and that the stalling mechanism is similar at different operating speeds.

The nature of the flow within turbomachinery is inherently viscous, unsteady and three-dimensional. The viscous effects can dominate if the casing boundary layer is much larger than the tip clearance with suction side flow separation at the blade tip resulting from the viscous
effects in the gap. These flow fields are extremely complicated and must be computed with accurate simulation tools if they are to be useful to designers exploring off-design conditions, as such an understanding of numerical solution behavior under these conditions is vital. The next section provides a look at approaches and challenges to modeling the tip clearance flow.

1.5 Tip Clearance Modeling

Accurate solution to the tip clearance flow in a compressor is one of the more difficult problems in numerical simulations because of the nature of the flow in this region. The three-dimensionality of the flow with no clear transition between the clearance flow and boundary layer on the endwall, the presence of a clearance flow vortex and the interaction of a shock with the boundary layer and clearance vortex exemplify the complexity of the flow. Linearized theories are unsatisfactory in computing this complex flow and complete Reynolds-Averaged Navier-Stokes equation based schemes are required to obtain a complete description of the flow. Many formulations are based on idealized geometries that do not incorporate the complex nature (geometry, adverse pressure gradient) of the boundary layers encountered and the inception of stall is applied empirically using such correlations as that by Koch [3]. While these methods do not incorporate the full details of the flow, acceptable predictions can be made for two-dimensional flows and isolated blade rows [14]. Historically, the clearance flow and its effect on the main flow have been modeled as an inviscid jet. A simple wedge shaped model of the blade tip has been used in place of the actual blade geometry to, in effect, artificially account for the blockage effects created by vena contracta [6]. Using a tip clearance smaller than the actual clearance level has also been somewhat effective in incorporating this effect. In addition to geometric tip modeling, grids also play an important role in predicting the structure of the clearance flow. Trends may be extracted using coarse grids and accounting artificially for blockage, but care must be taken to include enough grid points if the intent is to resolve the details of the clearance flow.
Identifying the point of stall and/or numerical breakdown in simulations as well as their cause is important to being able to resolve the flow accurately. Adamczyk, et al. [8] defined numerical onset of stall as the point of operation at which any increase in the static pressure leads to a steady decrease in the mass flow toward no convergence point at the inlet as the solution iteration count is advanced. As the flow rate is reduced, negative axial velocity in the tip region can be experienced and a recirculation may occur that spills forward of the leading edge. Copenhaver, et al. [6] and Adamczyk, et al. [8] encountered this type of recirculation in the tip near the leading edge and cited this phenomena as a possible cause of numerical (and perhaps physical) instability. A reduction in mass flow causes the leading edge shock to move toward the inlet and the leakage vortex to incline forward resulting in the location of the vortex/shock interaction and the blockage associated with it to move forward in the passage. This eventually leads to the spilling of the low-energy flow at the tip leading edge that causes an alteration in the upstream wave pattern, reduction in mass flow rate and increase in blade incidence. The larger incidence increases the leakage flow, pushing the shock forward and again leading to a steady reduction in mass flow.

1.5.1 Turbulence Modeling

Because of the nature of the flow in the tip region, which includes suction side separation, possible shock interaction and shear associated with the tip vortex, the type of turbulence model used can have a large impact on predictions in this region. While turbulent stresses in these situations are not sufficiently modeled using the conventional mixing length model nor a standard two-equation type model, a two-equation model has demonstrated improvements when low Reynolds number effects that can effect flow reattachment were included [15]. Near the stall point for a transonic compressor rotor, Chima [16] showed that a thick casing boundary layer, a strong vortex, wake, upstream flowing wall jet and shock induced separation were present and that inadequate prediction of these characteristics was determined.
to be the result of the turbulence models used (Baldwin-Lomax and single-length scale algebraic models). Turner and Jennions [17] showed variation in transonic fan predictions due to turbulence model and concluded that very successful results can be obtained using a combination of an explicit Navier-Stokes solver with an implicit $\kappa-\varepsilon$ model. While geometric tip model, grid and turbulence model have an effect on resolving the details of the clearance flow, trends and satisfactory results may sometimes be achieved using Navier-Stokes codes with coarse grids and unsophisticated turbulence models in the tip clearance region [16], further confusing prediction requirements.

1.5.2 Viscous Effects

The viscous effects associated with the tip clearance flow lead to blockage and influence the limiting pressure rise capability of the machine. The viscous effects present in shear layers both in the tip and the boundary layers (found in the casing and on the blade surfaces) combined with the presence of shocks are responsible for losses generated. Proper calculation of blockage and loss is necessary for accurate performance prediction. Currently, no general method exists for accurately predicting the blockage and loss making them the largest causes of prediction inaccuracy. In addition, smearing of the flow in large gradient regions results from artificial viscosity that is a consequence of finite difference approximations. While artificial viscosity may cause problems in the result, it can be useful in stabilizing the flowfield at the beginning of the calculations when large gradient transients may be present.

Performance predictions of compressor and fan rotors have improved dramatically over the past decade, however improvements are still required. Common CFD methods have great difficulty predicting the efficiency derivatives and stall range of compressor rotors at large clearances (>2% clearance-to-chord). Common CFD methods include steady isolated rotor analysis using structured block H-grids. The purpose of this investigation is to examine
clearance flow prediction approaches to suggest a strategy for determining criteria to improve clearance derivative predictions at large clearances.

The remainder of the thesis is laid out in the following manner. The next section provides a brief history of the stage examined in this paper. The numerical method and analysis techniques are then described, followed by a presentation of the numerical results and comparison with available data. Finally, conclusions are presented along with recommendations for further study.
Chapter 2

BACKGROUND ON WENNERSTROM DESIGN

This section presents the rotor examined in this paper. A brief history of the design intent of the rotor is followed by flow physics observed through previous experiments and numerical analysis. The rotor is representative of current design trends and has available data at multiple clearance levels. The experimental data have shown that this particular design is sensitive to tip clearance level and is a good candidate for tip clearance investigation.

The rotor examined in this paper is the transonic, high-throughflow, low aspect ratio design of Wennerstrom and Frost [18] The design originated in the development of an axial compressor to be used in a turbo-jet-type aircraft turbine engine. Objectives of the original design were:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure Ratio</td>
<td>12:1</td>
</tr>
<tr>
<td>Number of Stages</td>
<td>5</td>
</tr>
<tr>
<td>Overall Efficiency</td>
<td>84%</td>
</tr>
<tr>
<td>First Rotor Tip Speed</td>
<td>457 m/s</td>
</tr>
<tr>
<td></td>
<td>(1500 ft/s)</td>
</tr>
<tr>
<td>First Stage:</td>
<td></td>
</tr>
<tr>
<td>Pressure Ratio</td>
<td>1.912</td>
</tr>
<tr>
<td>Efficiency</td>
<td>83%</td>
</tr>
</tbody>
</table>
This design experienced transonic stall flutter that was later corrected using a ruggedized re-design, performed by Wennerstrom and Buzzel [19], that was able to maintain performance characteristics. Experimental first stage performance for the ruggedized design produced a pressure ratio of 1.93 and efficiency of 85.4 percent as shown by Law and Wadia [20]. This rotor is an example of the low aspect ratio, high throughflow design trend discussed earlier.

An unswept redesign of the Wennerstrom rotor (designated Rotor 4) resulted from a parametric design investigation meant to experimentally explore the effects of various geometric design factors. Parker and Simonson [21] provide details of this design. Copenhaver, et al. [15] investigated the ability of a steady state, three-dimensional Navier-Stokes method to predict the detailed flow features of this geometry at off-design conditions using isolated rotor simulations. While the predictions showed agreement with the experimental spanwise variations, loss calculations were not correct. Possible causes of these differences were numerical inaccuracies and the absence of a mixing model in the streamline projection to the rotor exit of the experimental stator exit data. The lack of a mixing model in the projections could cause level and radial distribution inaccuracies by not accounting for mixing losses. Although discrepancies were present, experimental and numerical results indicated that the midspan and hub regions showed higher efficiencies than the tip region, thus indicating the importance of understanding and controlling the flow near the endwall for high throughput, low aspect ratio, transonic rotors.

In addition to the Rotor 4 investigation, Copenhaver, et al. [6] investigated numerically the influence of tip gap size on an aft swept design (designated Rotor 10); another result of the parametric design investigation mentioned earlier. This geometry (high throughput, low aspect ratio, transonic rotor) was chosen for the current investigation because it is consistent with
current design trends and has been shown to be sensitive to tip clearance flow. Significant
design features of the rotor are:

Design Pressure Ratio 2.06
Number of Blades 20
Mean Aspect Ratio 1.32
Tip Speed 457.2 m/s  
(1500 ft/s)
Leading Edge Radius 148.66 mm  
(5.85 in.)

Copenhaver’s paper focused on the difference between two tip clearance levels (0.27  
and 1.87 tip clearance-to-chord ratios) and found that the aerodynamic behavior and mass flow  
rates for the different tip clearances were modeled reasonably well. Disagreements in the  
calculation of the aerodynamic loss and efficiency were observed and attributed to numerical  
deficiencies near the tip. Turbulence model and grid refinement were suggested as means for  
improvement. Performance reductions were found to result from increased clearance levels  
and were close to those mentioned previously by Smith [10]. Observations of the flow field  
depicted that the tip flow impacted mainly the outer 30 percent of span and tended to increase  
tip flow/shock interaction. The increased tip clearance level was also found to reduce the swept  
structure of the leading edge shock, indicating that the benefits of swept airfoils (to reduce  
shock losses) are reduced toward off-design conditions. Lastly, it was suggested that the tip  
clearance vortex/shock interaction plays a major role in understanding tip clearance flows and  
their impact on performance. These observations provide a comparison for flow field  
observations in the current study and indicate the need for improvement in performance  
predictions of the tip clearance flow.
It is important to note that tip clearance flows are not always detrimental to the rotor performance. Wennerstrom [7] observed an optimal rotor tip clearance during an experimental study of a high throughflow transonic axial compressor stage. Wennerstrom suggested that the effects of tip clearance flows along with secondary flows in the passage have a balancing point with the relative movement of the rotor and wall. The finding of an optimal clearance height in this instance indicates that one or the other of these effects dominate at clearances on either side of the optimum for the specific case analyzed. It is theorized that with smaller tip clearance the leakage vortex is not large enough to stabilize some of the secondary flows in the passage, causing excess blockage due to secondary flow. In contrast, a large clearance will dominate and cause excess blockage in the upper portion of the passage. While it is important to note that obtaining an optimum clearance is possible, it is usually out of the feasible operating range for the geometry and so, for most cases, expectations would be that any increase in tip clearance size will result in performance loss.

The previous studies mentioned above provide insight into the overall behavior of the tip flow and the impact of tip clearance level for the rotor considered in this study. While the numerical studies showed that overall performance can be calculated reasonably well, inadequacies in loss calculations were present. In all cases, the dominant influence on performance was the tip clearance flow. This indicates further improvement in numerical models and physical understanding is necessary to improve performance predictions. The current study investigates the impact of numerical approach (meaning multiple blade row, turbulence model and unsteadiness) on predictions of the flow field for a swept transonic compressor rotor with a large tip clearance in an effort to identify improved prediction methods and gain physical understanding of tip clearance flow. The next section presents the numerical method applied and analyses performed followed by analysis and discussion of the numerical results.
Chapter 3

MOTIVATION AND APPROACH BACKGROUND

Motivation for this study was prompted by the results of steady isolated rotor calculations using common grids and boundary conditions which resulted in premature stall of the rotor performance predictions at large clearance conditions. The grids are structured H-block type grids that incorporate the rooftop tip treatment described in Section 1.5. The number of grid points in each direction as well as in the tip is determined using a prescribed formula with the intent of providing balance between solution accuracy and computational time. The spacing near solid surfaces is chosen to allow for the use of wall-functions for boundary layer prediction. A two-dimensional throughflow code was used to determine inlet and exit radial profiles to prescribe the inlet and exit boundary conditions for the specified operating condition.

One concern was that the rotor back pressure profiles used from the two-dimensional throughflow analysis code could be in error and lead to excessively high tip loadings and premature stalling in the tip. A stage calculation would provide rotor radial exit static pressure profiles that include rotor-stator interaction effects. Figure 3.1 compares the exit static pressure profile predicted using the two-dimensional throughflow code and the time-averaged static pressure profile predicted from the unsteady stage simulation. The CFD code shifts the exit static pressure profile to obtain the desired one-dimensional exit corrected mass flow rate,
Predicted using 2-D Throughflow Code
Predicted using Stage Simulation
Figure 3.2. Calculated total pressure profiles at rotor exit at an exit corrected flow rate of 16.1 kg/s.
therefore the profile shape is being compared for illustrative purposes and the values have been excluded. The two-dimensional throughflow prediction was shifted to provide the same value at the rotor hub. The stage simulation resulted in a large gradient in the pressure profile near the tip when compared to the two-dimensional throughflow code.

The isolated rotor flow field was then calculated using both exit static pressure profiles to determine the influence of profile on the solutions. Figure 3.2 shows the resulting exit total pressure profiles, including experimental data. The total pressure profiles show a slight redistribution toward the tip as a result of the exit static pressure profile. However, as seen in the resulting total pressure profile (Figure 3.2), this had a minor effect on the average rotor pumping in the tip region. Therefore, the radial profiles differences alone do not explain the premature stalling dilemma. The maximum difference in the exit total pressure profiles is on the order of one percent which may be a result of the final exit corrected flow difference of 0.6 percent. The exit static pressure profile was concluded to have minimum effect on the predictions. The rotor exit static pressure profile from the time average of the stage calculation was then used as the exit boundary condition for the isolated steady and unsteady calculations for various operating conditions. In addition, isolated rotor analyses were completed using two different turbulence models, CMOTT $\kappa-\epsilon$ [22] and Wilcox $\kappa-\omega$ [23] with the Launder-Kato modification. These simulations were performed to explore the effects of these two models on the predicted loss and blockage in the tip region and its associated (or resultant) effect on the predicted performance characteristics of the rotor.

The goal of this research was to investigate clearance flow prediction effects of stator/rotor, steady/unsteady and turbulence model approaches with large tip clearance levels. The tip clearance examined was 1.96 mm giving a 1.87 percent tip clearance-to-chord ratio running at 100% design speed. The inadequacies experienced with standard design type CFD methods in predicting the efficiency derivatives and stall range of compressor rotors at large clearances causes a need for improved numerical methods and physical understanding of tip
clearance flow. Various simulations were performed for the purpose of investigating the range and limitations of numerical approaches on a swept transonic compressor rotor with large clearances. Table 3.1 provides a summary of these simulations.

Next, Chapter 4 discusses the numerical method used for the simulations. This includes presentation of the numerical algorithm, boundary conditions and computational grids used. This is followed with a presentation and discussion of the results in Chapter 5. Lastly, Chapter 6 provides conclusions and recommendations for further study.

<table>
<thead>
<tr>
<th>Blade Rows</th>
<th>Time Dependence</th>
<th>Turbulence Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stage</td>
<td>Unsteady</td>
<td>$\kappa-\varepsilon$</td>
</tr>
<tr>
<td>Isolated Rotor</td>
<td>Unsteady</td>
<td>$\kappa-\varepsilon$</td>
</tr>
<tr>
<td>Isolated Rotor</td>
<td>Steady</td>
<td>$\kappa-\varepsilon$</td>
</tr>
<tr>
<td>Isolated Rotor</td>
<td>Steady</td>
<td>$\kappa-\omega$</td>
</tr>
</tbody>
</table>

**Table 3.1:** Matrix of simulations performed.
Chapter 4

NUMERICAL METHOD

This section presents the numerical code, boundary conditions, grids and approaches used in the current studies. Three types of simulations were undertaken: steady isolated rotor, unsteady isolated rotor, unsteady stage. The boundary conditions and convergence criteria are described for all three approaches.

4.1 Algorithm

The computations were completed using the code MSU-TURBO, of Chen et al. [24,25], which solves the three-dimensional, Reynolds-averaged Navier-Stokes equations using a nonlinear, unsteady solution technique. The code is capable of handling a fully coupled stage configuration in addition to isolated blade rows. The spatial discretization of MSU-TURBO uses a third order upwind Roe flux-difference splitting and a minmod limiter for the inviscid fluxes. Viscous fluxes were evaluated with second order central differencing. The implicit temporal discretization is a Newton sub-iteration technique at each time step with a Gauss-Seidel matrix inversion procedure and is second order accurate in time. Both the CMOTT $\kappa-\epsilon$ [22] and Wilcox $\kappa-\omega$ [23] turbulence models were used with wall functions to resolve the flow along solid boundaries.

Steady-state convergence was reached when the overall mass flow histories reached a steady state and energy derivative residuals were reduced to less than $10^{-4}$. The derivative
residuals represent a change in the variable specified. As the mass flow was reduced and the performance moved closer to a stall situation, the mass flow history could not reach a steady state. In this situation, an average over one period of the cyclic state reached is provided. First order time accuracy and local time stepping convergence acceleration techniques were employed with steady calculations and resulted in computational time on the order of a few days.

There were two convergence levels: a) convergence of the subiteration process at each time step where average density and energy derivative residuals were reduced to less than $10^{-4}$ and b) global convergence for unsteady analyses was reached when the overall mass flow histories reached a cyclic state. CPU time on an HP11 platform for unsteady stage calculations were slightly more than four times that for an unsteady isolated rotor calculation with reduced computational time resulting from the use of parallel processing.

4.2 Boundary Conditions

Isolated rotor calculations used periodic flow conditions at periodic passage-to-passage interfaces. Total pressure, total temperature, inlet tangential flow angle and inlet radial flow angle were fixed and the one-dimensional Riemann invariant was extrapolated at the inlet. No slip, adiabatic wall conditions were applied on all solid surfaces. Steady exit flow conditions using prescribed pressure profiles were applied at the grid exit. Exit static pressure profiles were held using one-dimensional characteristic boundary conditions.

The stage simulations used steady inlet flow conditions with one-dimensional characteristic boundary conditions. No slip, adiabatic wall conditions were applied on all solid surfaces. Phase lag boundary conditions with a sliding interface were applied at the rotor/stator grid and periodic interfaces. Steady exit flow conditions prescribed pressure profiles at the stator exit.
4.3 **Computational Grids**

A single passage was used for both the stage and isolated rotor simulations. The stage calculations used a structured H-block grid with the rotor grid having 56 cells in the pitchwise direction, 68 cells in the spanwise direction and 160 cells in the streamwise direction and a stator grid with 57 cells in the pitchwise direction, 69 cells in the spanwise direction and 151 cells in the streamwise direction (Figure 4.1). The two grids interfaced at a point midway between the rotor trailing edge and the stator leading edge. Care was taken to ensure adequate leading and trailing edge resolution (Figure 4.2). The tip was modeled using a wedge with 16 cells located between the tip and the casing (Figure 4.3).

The same rotor grid used for the stage calculations was used for the isolated rotor calculations with a modification made to extend the exit plane further toward the stator leading edge and resulting in 57 cells in the pitchwise direction, 69 cells in the spanwise direction and 171 cells in the streamwise direction. Grid cell counts were selected based on experience.

The next section presents the numerical results and includes a description of the analyses performed. Overall performance is analyzed followed by a more in depth discussions on the influence of turbulence model and unsteadiness. Finally, conclusions are presented along with recommendations for further study.
Figure 4.1: Blade-to-Blade View of Stage Computational Grid

Figure 4.2: Rotor Leading and Trailing Edges

Figure 4.3: Computational Grid – for Wedge Tip Treatment
Chapter 5

NUMERICAL FLOW PREDICTION

This chapter presents the non-linear MSU-TURBO flow predictions for Wennerstrom rotor 10 with a 0.077-inch tip clearance. The chapter begins with a one-dimensional performance comparison of the four configurations (Table 3.1) presented in Section 5.1. The effects of turbulence model on the steady, isolated rotor simulations are examined in Section 5.2. The effects of unsteadiness on the isolated rotor predictions are compared in Section 5.3. The final section (5.4) examines unsteady analysis comparing coupled stage versus isolated rotor predictions.

5.1 One-Dimensional Performance Comparison

Performance maps including total pressure ratio, total temperature ratio and adiabatic efficiency across the rotor for all four approaches are compared to experimental results in Figures 5.1, 5.2 and 5.3, respectively, to evaluate the one-dimensional performance of the cases explored. Enthalpy averaged total pressure and mass averaged total temperature values were used in calculating pressure and temperature ratios. Enthalpy averaging was used when calculating the total pressure to ensure the inclusion of high loss regions. The performance map information was obtained using total pressure and temperature at each instant in time and time-averaged over the fundamental period of the unsteady flow, which was the frequency associated with the shock/vortex interaction for the “steady” solutions and the blade passing
Figure 5.1: One-Dimensional Performance – Pressure Ratio Map

ECF = Exit Corrected Flow Rate

Data – 0.279 mm Clearance
Data – 1.956 mm Clearance
Stage, Unsteady, κε
Rotor, Unsteady, κε
Rotor, Steady, κε
Rotor, Steady, κω
not fully converged

ECF = 16.1 kg/s
ECF = 16.4 kg/s
ECF = 15.7 kg/s
ECF = 15.2 kg/s

ECF = Exit Corrected Flow Rate
Figure 5.2: One-Dimensional Performance – Temperature Ratio Map

**INLET CORRECTED FLOW (kg/sec)**

**ROTOR TEMPERATURE RATIO**

Data – 0.279 mm Clearance
Data – 1.956 mm Clearance
Stage, Unsteady, $\kappa_e$
Rotor, Unsteady, $\kappa_e$
Rotor, Steady, $\kappa_e$
Rotate, Steady, $\kappa_w$
not fully converged

ECF = Exit Corrected Flow Rate
Figure 5.3: One-Dimensional Performance – Efficiency Map

INLET CORRECTED FLOW (kg/sec)

ROTOR ADIABATIC EFFICIENCY

ECF = Exit Corrected Flow Rate

Data - 0.279 mm Clearance
Data – 1.956 mm Clearance
Stage, Unsteady, \( \kappa_e \)
Rotor, Unsteady, \( \kappa_e \)
Rotor, Steady, \( \kappa_e \)
Rotor, Steady, \( \kappa_\omega \)
not fully converged

ECF = Exit Corrected Flow Rate
ECF = 16.1 kg/s
ECF = 16.4 kg/s
ECF = 15.7 kg/s
ECF = 15.2 kg/s

ECF = Exit Corrected Flow Rate

Data - 1.956 mm Clearance
Data - 0.279 mm Clearance
frequency for the unsteady simulation. Adiabatic efficiency was calculated using enthalpy averaged pressure and mass averaged temperatures (Equation 5.1). Inlet mass flow corrected to standard day conditions was used as the abscissa for the three curves. Experimental performance was obtained using total temperature and pressure measurements taken at the exit plane using vane-mounted probes. Smaller clearance (0.011 inch) data is included as a reference and to show the sensitivity of this geometry to tip clearance level. The inclusion of unsteady effects resulted in large static and total pressure gradients toward the tip leading to numerical breakdown in this region. As a result, much smaller time steps than expected were required to resolve the flow field for the unsteady calculations, thus increasing the computational time needed to reach a fully converged solution. Although these constraints prevented the lower flow unsteady numerical computations from reaching complete convergence in the data throttle range, the overall trends of the prediction approaches can be discerned and comparisons between approaches can be made.

Observing the one-dimensional performance maps shows the steady calculations underpredicting the temperature and pressure ratios when the $\kappa-\epsilon$ turbulence model is applied in addition to numerically stalling at a higher exit corrected flow than the data indicates. In contrast, the steady calculations with the $\kappa-\omega$ turbulence model begin to drop in pressure ratio at about the same exit corrected flow as the data indicates, but appears to grossly overpredict the performance characteristics. Extrapolating the trend of the unsteady simulations toward stall and comparing the steady and unsteady, $\kappa-\epsilon$ isolated rotor simulations show an improved throttle range and increased pressure ratio when unsteadiness is included in the computations.

$$\eta = \frac{(p_{0_{ex}}/p_{0_{in}})^{(\gamma-1)/\gamma} - 1}{(T_{0_{ex}}/T_{0_{in}}) - 1}$$  (5.1)
The unsteady stage calculations follow the unsteady isolated rotor performance at higher mass flow rates, but the pressure ratio falls earlier.

Vane-mounted probe measurements of the total pressure radial distributions and temperature were taken downstream of the rotor for the experimental peak efficiency at large clearance. These area-weighted circumferentially averaged experimental data are compared with each solution at a mass flow that most closely matches the experimental peak efficiency. The numerical solutions used for the comparison are indicated in Figure 5.4. Figure 5.5 shows the radial profiles of the predicted enthalpy averaged total pressure and mass averaged total temperature with the measured data. Again, the unsteady calculations provide the closest agreement with the data, with the unsteady stage calculation predicting the closest overall match with the data points. The unsteady, isolated rotor simulation offers a closer match to the total temperature profiles, but indicates overpumping near the casing when compared to measured total pressure profiles. Overpumping near the casing is observed with the steady $\kappa-\omega$ computational results and is investigated further in the next section. The steady $\kappa-\varepsilon$ and unsteady stage cases appear to be underpredicting temperature in the region just below the tip of the blade (70-90% span). The unsteady, isolated rotor calculation matches the data well for the 70-90% span range, however the temperature predictions are low in the upper 10% span and may be related to the slight over prediction of total pressure in the same range. The underpredictions may be due to insufficient mixing levels being modeled in this region resulting from turbulence model parameters or the effects of non-adiabatic wall conditions at the casing in the experiment.
Figure 5.4. Points used for comparison of total pressure and total temperature profiles downstream of the rotor at peak efficiency.
Figure 5.5. Comparison of total pressure and total temperature profiles downstream of the rotor at peak efficiency.
All four simulation configurations produced ‘unsteady’ output at lower flow rates. The inlet corrected mass flow histories for the steady, isolated rotor simulations presented in Figure 5.6 show oscillatory behavior that increases in amplitude as the mass flow is reduced. The $\kappa-\omega$ turbulence model shows a lower level of unsteadiness, but shows a more pronounced changing frequency content. The oscillatory behavior in the $\kappa-\omega$ histories is present at a higher flow rate. The $\kappa-\varepsilon$ steady and unsteady isolated rotor inlet corrected mass flow histories are then presented in Figure 5.7. The mass flow histories again show oscillatory behavior that increases in amplitude with reduced flow rate. The oscillations appear in the solutions at similar mass flow rates with similar levels with the unsteady isolated rotor showing unsteadiness at only a slightly higher mass flow. It should be noted that the ‘not fully converged’ solutions are showing variations of 1.63 and 1.91 percent for the unsteady, isolated rotor simulations and 6.50 and 9.64 percent for the stage simulations as the mass flow is reduced. The unsteady stage simulations produced the largest variation of the cases examined. An underlying frequency (below that of the blade passing frequency) begins to grow in amplitude as the cases progress toward lower flow rates. Figure 5.8 shows a plot of the inlet corrected mass flow throughout one period for two stage solutions. The frequency is observed to be lower than that of the stator blade passing frequency (requiring approximately 3.5 wheel revolutions for one period). This is suggestive of rotating stall within the annulus flow field but at a much slower rotation rate, perhaps attributable to the fact that these were single passage calculations. The oscillations present in the steady results indicates that steady flow solutions cannot be achieved under these flow conditions. In addition, the unsteady solutions exhibited not only blade passing frequency unsteadiness with amplitudes on the order of the steady case variations, but also lower frequency unsteadiness with much higher amplitude. The presence of the fluctuations in mass flow rate that do not correspond with the blade passing frequency in the stage calculations is commonly associated with rotating stall, but may be an indication of a precursor to a more
5.6: Steady, isolated rotor inlet corrected mass flow histories for three flow conditions.
5.7: Steady and unsteady $\kappa-\epsilon$ isolated rotor inlet corrected mass flow histories for three flow conditions.
5.8: Unsteady stage inlet corrected mass flow histories for two flow conditions.
established rotating stall, since the calculations do not reach the stall point, signifying the need for multi-passage analyses.

The overall results show the unsteady calculations providing the closest match to the data, however the largest variations in solution resulted from choice of turbulence model. A closer investigation into the unsteady stage prediction and the causes of the large variation in predictions due to steady analysis and turbulence model can provide some insight into the behavior of the tip flow. The next section examines differences in tip clearance flow associated with the two turbulence modeling algorithms and offers possible causes for the large differences in solution.
5.2 Turbulence Model Comparison

The previous section showed that the largest variation in solutions is the result of turbulence model. The steady results using different turbulence models straddle the unsteady solutions and the data. This section presents further details of the flow for the $\kappa-\varepsilon$ and $\kappa-\omega$ solutions to show how the difference in turbulent kinetic energy associated with each turbulence model offers an explanation of the differences experienced.

Comparison with the one-dimensional results show that the unsteady, isolated rotor calculation is consistent with the unsteady stage calculations, both of which use the $\kappa-\varepsilon$ turbulence model, but provides higher values than the steady $\kappa-\varepsilon$ solution. Observing that the unsteady isolated rotor and unsteady stage solutions perform consistently closer to the data than the steady results shows the effect of unsteadiness on the solutions. The variation in the steady solutions resulting from choice of turbulence model indicates the significant effect turbulence model has on the steady solutions particularly. Steady isolated rotor calculations were performed using the $\kappa-\varepsilon$ turbulence model and was found to stall prematurely. The $\kappa-\omega$ model was then applied to examine the impact of turbulence model on rotor performance and tip flow predictions and resulted in overpumping. The one-dimensional performance data presented in the previous section showed that the largest variation in results occurred due to choice of turbulence model. Both models began to show oscillations in the solution as the flow rate was reduced, however, the $\kappa-\varepsilon$ oscillations were larger in amplitude (see Figure 5.6). The fluctuations appear to correspond to the shock/vortex oscillation present in the solution; a known unsteady phenomenon. The presence of these cyclic fluctuations in the solutions demonstrates the presence of unsteady phenomenon and that the solution cannot be made to converge to a steady state. These
observations illustrate the need for unsteady calculations of the flow, especially at lower flow rates.

Further examination of the tip clearance flows obtained with the two turbulence models show the cause for the differences experienced. Comparisons are made between the \( \kappa-\varepsilon \) solution and the \( \kappa-\omega \) solution for the same exit corrected flow rates and includes a contour of the \( \kappa-\omega \) solution at its lowest flow rate. The points used for the comparisons are indicated in Figure 5.9. Mach contours of the two cases at higher and lower flow rates are compared to describe the flowfield at the tip and changes experienced as the rotor is throttled. A numerical simulation using the \( \kappa-\varepsilon \) turbulence model at an exit corrected flow of 15.1 kg/s resulted in numerical stall (as described in Section 1.5), thus no solution was obtained for comparison at this exit corrected flow rate. Figure 5.10 compares the relative Mach number contours for the \( \kappa-\varepsilon \) and \( \kappa-\omega \) solutions and Figure 5.11 indicates significant flow structures for an exit corrected flow of 16.1 kg/s. A solution for each modeling case at a higher and lower exit corrected flow condition is included in the comparison to show the changes in the tip flow as the rotor is throttled toward stall. Calculations of the shock position and angle are identical at higher exit corrected flow (16.1 kg/s) and begin to move forward in the passage as the flow rate is reduced (see Figures 1.9 and 1.10). Notable differences are that the shock in the \( \kappa-\omega \) solution impinges on the suction surface and the shock begins to move toward the inlet at a higher flow rate for the \( \kappa-\varepsilon \) solutions. Although both turbulence models show an increase in low energy fluid (relative to the main flow) within the passage as the mass flow is reduced, the levels of low energy fluid are higher for the \( \kappa-\varepsilon \) solution. The tip vortices have similar strength and trajectory at the leading edge of the blade, but differ as they move further into the passage. The \( \kappa-\varepsilon \) vortex does not travel as far into the passage and has much larger gradients where it meets the main flow and shock and spreads closer to the leading edge at a higher rate indicating that it is spreading and dissipating more quickly.
Figure 5.9: Pressure ratio map indicating turbulence model comparison operating points.
Figure 5.10: Relative Mach contours at blade tip for steady, isolated rotor simulations.
Figure 5.11: Relative Mach contours at blade tip for steady, isolated rotor simulations indicating flow structures.
These observations are consistent with the one-dimensional results and show the larger blockage (up to 35% more blocked area in the tip) associated with the $\kappa-\varepsilon$ model that produces the lower speedline seen in Figures 5.1, 5.2 and 5.3.

The leading edge vortex is investigated further using streamlines from the tip leading edge (Figure 5.12) at the same operating points as presented in Figure 5.9. The vortices for all cases appear to have a tight core and follow the same path to about near the leading edge at the highest exit corrected flow rate (16.1 kg/s) with the $\kappa-\omega$ model retaining a tight vortex through the passage and the $\kappa-\varepsilon$ solution dissipating at about 50% chord. As the flow rate is reduced, the $\kappa-\varepsilon$ vortex structure becomes looser and spreads more quickly at the point of shock interaction. The $\kappa-\omega$ vortex also shows a less defined structure past the point of shock interaction as the mass flow is reduced. This looser structure of the vortices at the lowest flow rates may signal the onset of vortex breakdown and could be a contributing factor to stall. The $\kappa-\omega$ solution showed the formation of another vortex-like structure at about 50% chord that forms from clearance flow that enters the passage downstream of the leading edge. Figure 5.13 examines the vorticity associated with the tip flow for a closer investigation of the relationship between the two vortices. For both high and low flow cases, the vorticity in the $\kappa-\varepsilon$ solution reduces more quickly as it continues downstream both in the vortex core and the shear jet associated with the leakage flow. The second vortex structure observed in the $\kappa-\omega$ cases has vorticity counter to that of the leading edge vortex. This secondary vortex appears to become more intense as the leading edge pressure side of the adjacent blade row approaches the leading edge vortex. The induced vortex appears to be caused by a wall-bounded shear layer that exists as a result of a velocity difference between the shroud and leakage jet (see Figure 5.14) similar to that described by Van Zante, et al. [26]. Reduced diffusion would allow for a tighter tip vortex in the $\kappa-\omega$ case that remains strong further into the passage. With the vortex remaining stronger as it progresses into the
Figure 5.12: Tip vortex streamlines for steady, isolated rotor simulations.

A. STEADY, ISOLATED ROTOR, $\kappa-\omega$

B. STEADY, ISOLATED ROTOR, $\kappa-\epsilon$
Figure 5.13: Vorticity contours for steady, isolated rotor simulations.
Figure 5.14: Vorticity contours and velocity vectors taken perpendicular to tip flow vortex for steady, $\kappa-\omega$ isolated rotor simulation at 15.1 kg/s exit corrected mass flow.
passage, it meets the shear layer of the approaching blade creating the induced vortex. The absence of the secondary vortex in the $\kappa-\varepsilon$ cases suggests that larger turbulent viscosity and energy exist in the tip region.

To further explain the differences seen in the tip flow between the turbulence models, the turbulent viscosity at the blade tip for the same operating conditions is shown in Figure 5.15. The most striking difference is the large turbulent viscosity at the entrance and exit of the $\kappa-\varepsilon$ computational space with the $\kappa-\varepsilon$ solutions having levels more than two orders of magnitude larger than that the $\kappa-\omega$ solutions. The large levels of turbulent viscosity cause the tip flow to mix and diffuse at a higher rate in the $\kappa-\varepsilon$ solution leading to a looser tip vortex that diffuses early in the passage as seen previously.

The structure of the flow fields produced by the two turbulence models is quite striking. Figure 5.16 shows the turbulent kinetic energy (TKE) for both turbulence models at the high and low flow conditions. The $\kappa-\varepsilon$ solution shows a larger region of high TKE that spreads farther across the passage and is shifted forward in the passage as the flow rate is reduced. High values of TKE encompass the passage from blade to blade and leading to trailing edges in the $\kappa-\varepsilon$ solution. The large amount of turbulent energy resulting from the $\kappa-\varepsilon$ model indicates increased losses due to mixing. The blockage associated with the large region of low-energy flow observed in the Mach contours (Figure 5.10) combined with the increased mixing losses seen as a result of turbulent model result in lower performance characteristics for the $\kappa-\varepsilon$ model.

The differences observed in the steady solutions as a result of turbulence model illustrate the effects this parameter can have on the numerical predictions. Having shown that the unsteady isolated rotor solution provides a one-dimensional performance consistent with the trend of the data and the unsteady stage solution indicates further that the choice of turbulence model has a greater impact on the steady simulations. The presence of time-
varying fluctuations in the steady solutions as the mass flow is reduced suggests the presence of an unsteady phenomenon and that the flow cannot be made to converge to a steady state, as the flow rate is reduced. The next section looks into the details of the unsteady stage solution. The nature of the unsteady solution will provide insight into the physics of the unsteadiness observed in the flow.
Figure 5.15: Turbulent viscosity contours at blade tip for steady, isolated rotor simulations.
Figure 5.16: Turbulent kinetic energy contours at blade tip for steady, isolated rotor simulations.
5.3 Steady/Unsteady Comparison

This section compares the isolated rotor results for the $\kappa-\epsilon$ turbulence model to examine the effect of unsteadiness on off-design performance predictions. The steady results presented earlier underpredicted pressure and temperature ratios while the unsteady solutions appear to agree better with the data. It can be inferred from the one-dimensional analysis that the lower order temporal accuracy of the steady calculation results in greater damping of the solution, and hence delays the onset of unsteadiness at low flow rates. The one-dimensional performance data presented earlier showed that the largest variation in results occurred due to choice of turbulence model, however the variation in performance seen by the use of steady or unsteady analysis is also significant. Again, the steady solutions began to show unsteadiness as the flow rate was reduced which is consistent with the observations of the previous section that relate the fluctuations to the unsteady shock/vortex oscillation present in the solution. The presence of these cyclic fluctuations in the steady solutions illustrates that the unsteady phenomenon present at the lower flow rates is beginning to dominate the flow and that the solution cannot be made to converge to a massflow absence of periodic fluctuations. These observations support the previous inference that unsteady calculations of the flow are needed, especially at lower flow rates. Further examination of the tip clearance flow differences between the two methods shows the impact time-accurate modeling on tip flow and compressor stall margin predictions.

Comparisons are made between three exit corrected flow rates (see Figure 5.17) as the rotor is throttled to show the resulting flow behavior of each approach as it moves toward stall. Figure 5.18 compares the relative Mach number contours for the steady and unsteady isolated rotor solutions with time-averaged conservative flow fields used to compare the solutions. The steady and unsteady isolated rotor solutions produced virtually identical results at an exit corrected flow of 16.4 kg/s. As the exit corrected flow was reduced to 16.1 kg/s, slight differences are seen with the shock and the tip leakage vortex pushing slightly more forward.
Figure 5.17: Pressure ratio map indicating steady/unsteady isolated rotor comparison operating points.
Figure 5.18: Relative Mach contours at blade tip for isolated rotor, steady and unsteady time-averaged simulations.
and producing a larger region of low energy flow in the passage in the steady solution. The steady solution begins to see a large drop in performance as the exit corrected mass flow is reduced to 15.7 kg/s with the Mach contours showing a dramatic difference in the tip flow. The shock is pushed forward of the passage with the tip vortex spreading and the entire passage encompassed by low energy flow with a large gradient seen at the entrance to the passage that is almost perpendicular to the axial flow direction. This signifies a large blocked area leading to the reduction in the one-dimensional performance seen by the steady calculations.

The differences in the tip flow are examined further in Figure 5.19 using axial velocity contours at the blade tip for the same exit corrected flows presented in Figure 5.17. The contours show negative axial velocities, representing reverse flow associated with the tip clearance spillage and subsequent vortex. At exit corrected flow of 16.1 kg/s the steady calculation shows the tip vortex not progressing as far into the passage as the unsteady calculation and curving toward the passage entrance. At an exit corrected flow of 15.7 kg/s, the steady calculation shows a large area of reverse flow dominating the passage at the rotor tip. This reverse flow shows flow spilling across the tip gap that is spreading rapidly from the blade leading edge that may be the result of increased numerical diffusion generated by the use of a larger time step in the steady calculations. The unsteady calculations indicate a much stronger, tighter, and more stable vortex at this flow rate and explains why the unsteady simulations are able to throttle to lower mass flows. The increase area of negative axial velocity in the passage at the tip further shows the impact of the blockage (200% more at an exit corrected flow of 15.7 kg/s) created by the tip clearance for the steady simulations.
Figure 5.19: Contours of negative axial velocity at blade tip for isolated rotor, steady and unsteady time-averaged simulations.
Figure 5.20: Mass flow history for steady and unsteady, isolated rotor simulations at exit corrected flow of 15.7 kg/s.

Figure 5.21: Instantaneous contours of negative axial velocity at blade tip for isolated rotor, steady and unsteady simulations at exit corrected flow of 15.7 kg/s.
As mentioned previously, unsteadiness was seen in both the unsteady and the steady calculations at reduced mass flow (Figure 5.20). The variations present at this reduced mass flow are presented in Figure 5.21 again using negative axial velocity contours. The figure presents instantaneous views of the tip flow at maximum, mid and minimum flow rate variation present in the solution. These contours show the reverse flow associated with the tip clearance flow moving forward and back in the passage and beginning to spill forward of the leading edge at the minimum flow. The low energy region moves forward in the passage, increasing the blockage then moves away from the leading edge, reducing the blockage. The unsteady solution appears to resolving details of the vortex shedding (or breaking away) in time whereas the steady solution shows a smeared flow that oscillates through the iterations. The flow in the steady solution has a much lower energy associated with it that provides an explanation to the lower performance seen in the pressure and temperature ratio plots presented in Section 5.1.

The blade incidence is visualized in Figure 5.22 using relative flow angle (measured from the axial direction). These figures show the angle of incidence at the blade leading edge decreasing (minimum and mid2) and increasing (maximum and mid1) relative to the blade through the oscillation. During a period of decreased incidence, the flow experiences a temporary stall, such as that presented in Figure 1.2. These observations suggest that the flow may be experiencing a stall / recovery pattern similar to rotating stall.

As mentioned in Section 1.5.2, improper calculation of blockage and loss can lead to inaccurate performance predictions. Denton [25] presented entropy creation as ‘the only rational measure of loss in an adiabatic machine.’ For this reason, entropy contours are presented in Figure 5.23 to examine the loss present in the steady and unsteady isolated rotor calculations in an effort to explain the differences present in the solutions. The solutions are shown again for the exit corrected mass flow rates presented in Figure 5.17. These contours show the entropy generation in the steady calculations to increase at a much greater rate as the mass flow is reduced, leading to the premature drop in performance. Denton [25] pointed out
Figure 5.22: Instantaneous contours of relative flow angle measured from axial direction at blade tip for isolated rotor, steady and unsteady simulations at exit corrected flow of 15.7 kg/s.
that high rates of entropy generation are likely to be found in regions where high velocities coincide with high viscous forces, such as the tip clearance flow, and that the mixing processes that take place between the tip leakage and main flows are the largest generators of entropy in this region. From observing the contours in Figure 5.23, it appears that the steady calculations are overpredicting the entropy generation, hence loss associated with the tip leakage flow. This could possibly be explained by the large time step that results from the use of first order time accuracy and local time stepping for steady calculations leading to reduced resolution of the flow field.

The results presented in this section show that the solution contains unsteady phenomena that cannot be converged to a steady solution as the mass flow is reduced, with the variations seen in the lowest flow solutions suggesting that a rotating stall type phenomenon may be present, although it is speculated that single passage calculations do not represent the phenomena with sufficient fidelity. The solutions also demonstrate that the steady calculations overpredict entropy generation in the tip region leading to the premature drop in performance resulting in an extended throttle range with the inclusion of unsteady effects. The differences seen in the two solutions may be a result of the lower order temporal accuracy in the steady solutions or the idea that absence of a true temporal term in the steady calculations prevents proper numerical convection that would allow for the recovery pattern found in the unsteady solutions. The reduced influence of the convective terms may not allow the solution to fully ‘recover’ during the oscillations in mass flow seen in Figure 5.20. These differences in method lead to the significant change in flow structure observed. The next section looks further into the unsteady stage calculation in an effort to provide insight into the effects of rotor/stator interaction on performance predictions.
Figure 5.23: Entropy contours at blade tip for isolated rotor, steady and unsteady simulations.
5.4 Stage/Isolated Rotor Comparison

The intent of the numerical approach study was to determine the strategies that show improvement in performance predictions and stall range at large clearances. The unsteady stage calculations provided the closest matches to the radial profile data (Figure 5.5), therefore details of the solution are investigated further to provide insight into the nature of the tip clearance flow for this configuration. The following examinations of the unsteady stage flow field will first compare calculated to measured static pressure contours the predictions. A comparison will then be made between the tip flow structure at lower (exit corrected flow of 15.7 kg/s) and higher mass flows (exit corrected flow of 16.1 kg/s), shown in Figure 5.24 to correlate observations made by Copenhaver, et al. [14] with this geometry. The overall nature and level of the unsteadiness produced through these simulations will then be analyzed, followed by a look at the instantaneous behavior of the flowfield as it oscillates through unsteady variations. The ensuing discussion will show that the unsteady stage simulations provide reasonable agreement with measured values. These observations support the solutions obtained and provide the suggestion that a rotating stall type phenomenon may exist.

Before showing these comparisons it is important to again point out that the unsteady solutions exhibit two distinct oscillation frequencies along the speed line. At high throughflow the oscillation is essentially blade passing frequency, whereas this is augmented by a low frequency oscillation reminiscent of rotating stall as one moves up the speed line. The flow structure near the casing is compared with measured values using casing static pressure contours shown in Figure 5.25. Due to the large variations in massflow, a solution locally time-averaged over one blade passing frequency about a mean value of the flow range for the unsteady stage is used for comparison with measured values. The measured and simulated contour plots have exit
Figure 5.24: Pressure ratio map indicating stage comparison operating points.
Figure 5.25: Measured and unsteady stage simulation casing static pressure contours near peak efficiency.
corrected flow values within 2.5 percent. Allowing for the difference and in exit corrected mass flow between measured and calculated combined with observed variations in the mass flow of the numerical solution (on the order of 1.75 percent), the comparison is sufficient to show that the flow structures are being correctly modeled. The measured shock appears less developed than that obtained in the simulations. This would be expected since the measured value is at a slightly higher throttle, although the observable area of the measured flow field leaves room for interpretation, as does the accuracy of the measurement technique. Both calculated and measured values show a shock structure detached from the blade with the simulations showing a large stagnation region at the blade leading edge. The shock is pushed downstream on the suction surface indicating a strong interaction with the tip leakage vortex although this interaction appears less intense in the measured values. These mild differences are accounted for by the difference in flow rate (thus, operating condition). The influence of the vortex and shock are both suggested by the regions of low pressure extending from the leading edge into the passage and forward of the passage, respectively.

Shock/vortex interaction is further examined in Figure 5.26 with time averaged relative Mach contours at the blade tip compared at exit corrected flow values of 15.8 and 16.1 kg/s (Figure 5.24). Both cases show a region of low energy fluid downstream of the vortex/shock interaction. The vortex is indicated by the two regions of high gradient passing diagonally through the passage centered on the pressure surface leading edge. The gradient toward the passage entrance curves as it nears the pressure surface. This curvature is the point where the shock intersects the leakage vortex and indicates a spreading of the vortex. This region of low energy fluid enlarges and moves forward in the passage as the mass flow is reduced. The observations made thus far are similar to those made in previous tip flow studies [1,4,11,14]. Having shown the unsteady stage calculations are in relative agreement with the measured data and correspond with the tip flow behavior expected as the rotor is throttled toward stall, the unsteady nature of the clearance flow is examined next.
Figure 5.26: Relative Mach number contours at blade tip for low and high flow unsteady stage simulations.
Figure 5.27: Inlet corrected mass flow history.
The unsteadiness in the stage analysis flow solutions increased as the solutions progressed up the speedline toward higher pressure ratios. The largest variation was observed for the peak pressure ratio obtained (exit corrected flow rate of 15.8 kg/s). Figure 5.27 shows that the inlet corrected flow for this solution showed overall variations of close to 2 percent. The overlying frequency is much lower than the blade passing frequency (estimated to be about 70 stator blade passing periods) with the solution requiring 6 periods of this lower frequency to reach a cyclic stage at constant amplitude. This low overlying frequency increases in amplitude as the rotor is throttled and suggests the presence of either a rotating stall or surge-like phenomena that increases as the mass flow is reduced. This is significant in that it may indicate signs of the onset of stall and with further studies including multiple passage simulations could provide insight into the nature and causes of rotating stall; alternatively this could be the result of the exit corrected flow boundary condition applied at the stator exit. The discussion concludes by observing the instantaneous variations in the tip clearance flow experienced through the mass flow variations.

Changes in the exit total pressure profile through one period of the overlying low frequency mentioned in the previous paragraph are shown in Figure 5.28. The instantaneous profiles are taken at the maximum, mid1 and minimum flow rates encountered through one period, as shown in Figure 5.27. The pressure oscillations indicate that the exit flow experiences redistribution as it passes from the maximum to minimum flow rate. The total pressure at the tip fluctuated approximately 6084 Pa (0.88 psi) throughout one period. Figure 5.29 compares the calculated total pressure profiles downstream of the stator trailing edge with measured values obtained using rakes downstream of the stator. The energy-averaged total pressure profile for a stator solution time-averaged across one blade-passing period at mid-cycle is used as the averaged value along with three instantaneous flow files that encompass
Figure 5.28: Rotor exit total pressure contours for the low flow unsteady stage calculation for the full range of exit corrected flow values.
Figure 5.29 Comparison of total pressure and total temperature profiles downstream of the stator at peak efficiency for the low flow unsteady stage calculation for the full range of exit corrected flow values.
the flow range. Fluctuations in total pressure are about 22 percent of the level seen downstream of the rotor (1845 Pa / 0.26 psi). This is not surprising due to the larger values of unsteadiness expected to be found near the rotor wakes. Interestingly, here the total pressure at the maximum flow agrees with the data well above 70 percent span while the total pressure for the minimum and maximum flows show the best agreement in the lower 40 percent span. The total temperature profile shows over-prediction of the temperature above 80 percent span. The same trend was observed previously downstream of the rotor and may be due to the adiabatic wall boundary conditions at the casing.

Figure 5.30 shows instantaneous relative Mach number and negative axial velocity contours over the flow range to explore the nature of the tip clearance flow as it passes from the maximum to minimum flow rate. The relative Mach contours show a high gradient region emanating from the leading edge and running diagonally into the passage counter to the rotation of the blade, signifying the core of the tip leakage vortex. A leading edge shock standing ahead of the blade interacts with the shock/vortex interaction indicated by the large area of low energy fluid. This point of interaction moves closer to the suction surface as the mass flow drops. This probably occurs as the result of a slightly weaker tip flow and shock that would be expected to occur at a lower flow. As the flow drops, the lower energy fluid moves forward in the passage and the core of the vortex becomes less curved away from the suction side. Downstream of the point of shock interaction, the vortex is severely weakened and the low momentum fluid begins to dominate the passage with reduced flow. Indications of flow pushing forward in the passage are reinforced with the plots of negative axial velocity. The reverse flow region associated with the tip leakage flow moves forward in the passage. The leakage flow moves diagonally across the passage impinging upon the tip region of the adjacent blade row and spilling over into the adjacent blade row downstream of the leakage vortex. The unsteady calculation provides a visualization of the flowfield during a period corresponding to
Figure 5.30: Relative Mach number and negative axial velocity variations at the blade tip for the low flow unsteady stage simulation.
the overlying low frequency that exists in the flow and provide indications of the presence of an instability or rotating stall-like phenomenon.

Examination of the unsteady stage calculation has demonstrated the complex nature of the tip clearance flow and demonstrates the need for performing unsteady analysis. Observations of the tip flow cycling in strength through the unsteady fluctuations suggest that either the blockage associated with a stalled cell or reverse flow associated with surge is present. The determination of exactly the type and nature of the phenomena is beyond the scope of this paper, however future investigations involving multiple blade passages to examine further the unsteady behavior are planned. The inclusion of multiple blade passages in a numerical simulation would allow the examination of passage to passage variations in the flow field that are necessary to identify the exact nature of the unsteady phenomena.
The tip clearance flow for the Wennerstrom Rotor 10 was simulated using four numerical approaches: 1) steady, isolated rotor with $\kappa-\varepsilon$ turbulence model, 2) steady, isolated rotor with $\kappa-\omega$ turbulence model, 3) unsteady, isolated rotor with $\kappa-\varepsilon$ turbulence model and 4) unsteady, stage $\kappa-\varepsilon$ turbulence model. The purpose of the investigation was to investigate clearance flow prediction approaches in an effort to suggest a strategy for determining criteria to improve clearance derivative predictions at large clearances. This chapter presents a summary of the findings and conclusions as well as recommendations for further study.

- The unsteady solutions provided the closest fit to the trend of the data. The unsteady isolated rotor simulations provided the closest one-dimensional performance match with the data. The unsteady stage simulations provided the closest exit radial profile data match and proved to be computationally expensive.

- The appearance of unsteadiness in all simulations as the flow was reduced indicates the need for unsteady calculations of the flow. The increase in unsteadiness with a reduction in flow rates that was observed shows that using unsteady calculations becomes more important as the rotor is throttled.
• The inability to reach a steady converged solution at lower flow rates indicates the presence of unsteady flow phenomena that cannot be simulated in steady state, further supporting the need for unsteady simulations at off-design conditions.

• The large variation in solution due to turbulence model indicates the need for more accurate turbulence modeling in the tip region, particularly for steady simulations.

• The cyclic behavior of the unsteady stage calculation during throttle suggests the possible presence of a rotating stall like phenomenon. Multiple passage simulations are suggested for further investigation into this phenomenon to gain insight into its nature and its influence on numerical modeling.
REFERENCES


