Impact of Unsteady Flow Processes on the Performance of a High Speed Axial Flow Compressor

by

Barbara Brenda Botros

B.Eng. Honours, Mechanical Engineering, McGill University, Montreal, Quebec, Canada, 2005

Submitted to the Department of Mechanical Engineering
in partial fulfillment of the requirements for the degree of

Master of Science in Mechanical Engineering

at the

Massachusetts Institute of Technology

February 2008

January 2008

©2008 Massachusetts Institute of Technology. All Rights Reserved

Author: ___________________________ ___________________________ Department of Mechanical Engineering

January 18, 2008

Certified by: ___________________________ ___________________________ John J. Adamczyk

Distinguished Research Associate, NASA Glenn Research Center

Thesis Supervisor

Certified by: ___________________________ ___________________________ Edward M. Greitzer

H. N. Slater Professor of Aeronautics and Astronautics

Thesis Supervisor

Certified by: ___________________________ ___________________________ Choon S. Tan

Senior Research Engineer

Thesis Supervisor

Read by: ___________________________ ___________________________ John G. Brisson II

Professor of Mechanical Engineering

Accepted by: ___________________________ ___________________________ Lallit Anand

Chairman, Department Committee on Graduate Students
DISCLAIMER OF QUALITY

Due to the condition of the original material, there are unavoidable flaws in this reproduction. We have made every effort possible to provide you with the best copy available. If you are dissatisfied with this product and find it unusable, please contact Document Services as soon as possible.

Thank you.

The images contained in this document are of the best quality available.
Impact of Unsteady Flow Processes on the Performance of a High Speed Axial Flow Compressor

by

Barbara Brenda Botros

Submitted to the Department of Mechanical Engineering on January 22, 2008 in partial fulfillment of the requirements for the degree of Master of Science

Abstract

This thesis examines the unsteady interactions between blade rows in a high Mach number, highly-loaded compressor stage. Two straight vane/rotor configurations with different axial spacing between vane and rotor are considered. The numerical simulations of the two configurations are used to determine the effect of axial blade row spacing on the level of entropy generation and the flow mechanisms that affect stage performance. The rotor shock waves that impinge on the upstream blade row result in shed vortices that convect downstream through the rotor. At the reduced axial spacing, vortices with larger circulation and entropy are formed.

Local entropy generation is assessed using a new numerical technique that allows adequate evaluation of spatial derivatives in high gradient regions, such as shock waves. It is found that the main difference in entropy generation between the two configurations studied is associated with the shed vortices. Entropy production and rotor work input depend on the vortex trajectory within the rotor, which in turn depends on the ratio of time scales: the time for vortex convection between blade rows, and the rotor period (i.e. the time for the rotor to move one rotor pitch), for a fixed geometry and inlet Mach number.

Thesis Supervisor: John J. Adamczyk
Title: Distinguished Research Associate, NASA Glenn Research Center

Thesis Supervisor: Edward M. Greitzer
Title: H.N. Slater Professor of Aeronautics and Astronautics

Thesis Supervisor: Choon S. Tan
Title: Senior Research Engineer
Acknowledgements

I am grateful to all the people who made this work possible. My advisors, Prof. Edward M. Greitzer and Dr. Choon S. Tan, have given me the opportunity to explore a research problem without reserve, to be able to make mistakes and learn from them. They have taught me a great deal about the research and writing process that has made this experience invaluable. Like vorticity, my gratitude to them is endless. To Dr. John Adamczyk, a great mentor and friend, my sincerest and warmest thanks for all he has taught me. With every ten hour tutorial and every cup of tea, he encouraged me to be better and work harder.

This thesis would not have been possible without the help of Sean Nolan, who ran the numerical simulations. I appreciate his guidance and patience throughout this work. I would also like to acknowledge the constructive input from Dr. Steven Gorrell from Air Force Research Laboratory. My gratitude goes to Dr. J.P. Chen, from Ohio State University, for his assistance using MSU Turbo. I also benefited from discussions with Prof. Nick Cumpsty from Imperial College, England about my work.

Many thanks to the students in the Gas Turbine Lab who made this experience that much more enjoyable. Alfonso Villanueva, mainly for his visualization codes, but also for his countless distractions. My right hand, Francois Lefloch, for being the best tutor, director, sounding board, tour guide, and co-member of the Terrible Two I’ve ever had. Darius Mobed, comedic genius and most potential castaway. Honorable mentions go to Mikhael Krotov (best office mate), David Tarr, Angelique Plas, Parthiv Shah, Ryan Tam, and Isabelle Delefortrie for the good times.

I could not have done this work without the support and encouragement from family and friends. I would like to credit my parents for giving me the perfect name – my initials played nicely into this thesis. I would especially like to thank my dad, Dr. Kamal K. Botros, for being an endless source of guidance and wisdom, for his encouragement, and for being the ultimate B.F.I.T.

This work was supported by AFOSR grant #FA9550-05-1-0050, under program managers Lt. Col. Rhett Jeffries and Dr. Thomas J. Beutner. Additional backing was provided by NSERC Canada Postgraduate Scholarships. This support is greatly appreciated. Numerical simulations were conducted using MSU Turbo.

Disclaimer

The research reported in this document/presentation was performed in connection with the U.S. Air Force Office of Scientific Research, USAF under grant number FA9550-05-1-0050. The views and conclusions contained in this document/presentation are those of the authors and should not be interpreted as presenting the official policies or position, either expressed or implied, of the U.S. Air Force or the U.S. government unless so designated by other authorized documents. Citation of manufacturers’ or trade names does not constitute an official endorsement or approval of the use thereof. The U.S. Government is authorized to reproduce and distribute reprints notwithstanding any copyright notation hereon.
List of Figures

1.1 Effect of axial blade row spacing on performance of a 4-stage compressor [1]......12
1.2 Comparison of wave configurations between the closest and farthest blade row
    spacing used in the SMI rig........................................................................13
1.3 Rotor shock impinging on the upstream vane row causing a net loading on the blade,
    and the formation of shed vortices.............................................................15
1.4 Isentropic efficiency as a function of axial spacing for transonic operating
    conditions [11]............................................................................................17
2.1 Vane/rotor-only performance from experiment, 100% speed. [21] ..................27
3.1 Entropy flux as a function of axial location for far and close spacing...........32
3.2 Entropy contours at mid-span for far and close spacing............................34
3.3 Divergence of velocity contours at mid-span for far and close spacing.........35
3.4 Control volume around rotor shock wave..............................................36
3.5 Comparison of entropy rise calculated from the computational and physical
    dissipation in axial gap for far spacing........................................................39
3.6 Isolating rotor shock waves. Contour plots of divergence of velocity and
    computational dissipation at the mid-span for the far spacing ....................40
3.7 Profile of divergence of velocity in shock region......................................41
4.1 Different vortex trajectories within the rotor passage due to different axial blade
    row spacing..................................................................................................45
4.2 Entropy contours for far2D and close2D.....................................................49
4.3 Time-averaged entropy contours within the rotor passage for far2D and close2D....50
4.4 Vortex equivalent radius.................................................................................. 52
4.5 Non-uniformity metric as a function of axial position................................. 55
4.6 Differences in boundary layers between far$_{2D}$ and close$_{2D}$.................. 58
B.1 Mixing of a swirling flow to a uniform flow state........................................ 74
List of Tables

2.1 Number of time steps per period used in Turbo code.................................24
2.2 Axial blade row spacing for close and far configurations, normalized by the axial rotor chord length $c_x$.................................................................25
2.3 SMI Aerodynamic Design Parameters [6].....................................................26
2.4 Number of grid points in axial, radial and pitch-wise directions for the three-dimensional calculations.................................................................26
2.5 Mass flow rate, total temperature rise, total pressure ratio, and adiabatic efficiency from three-dimensional calculations for far and close spacing configurations.....28
2.6 Number of grid points in axial, radial and pitch-wise directions for the two-dimensional calculations.................................................................29
2.7 Mass flow rate, total temperature rise, and total pressure from two-dimensional calculations for far$_{2D}$ and close$_{2D}$ configurations........................................30
2.8 Efficiency for far$_{2D}$ and close$_{2D}$ spacing configurations at the rotor exit and at downstream infinity.................................................................30
3.1 Difference in entropy generation within the rotor for the far and close spacings separated into the differences inside and outside the masked (shock) region.......43
3.2 Results from Table 3.1, excluding the end wall region....................................43
4.1 Circulation for the clockwise vortex at the rotor trailing edge......................51
4.2 Circulation-weighted radii for the clockwise vortex at the rotor trailing edge....52
4.3 Time-averaged entropy generation in specified control volumes within rotor.....57
4.4 Entropy rise from the rotor trailing edge to far downstream..........................60
Nomenclature

Symbols

\( \rho \) density

\( \rho \) pressure

\( \dot{m} \) mass flow rate

\( \eta \) adiabatic efficiency

\( \omega \) vorticity

\( \Gamma \) circulation

\( L \) axial gap spacing

\( c_s \) axial rotor chord length

\( \lambda \) trailing edge blockage

\( \phi \) shock angle

\( M_{rel} \) relative Mach number

\( s \) entropy

\( T \) temperature

\( p_i \) stagnation pressure

\( T_i \) stagnation temperature

TR stagnation temperature ratio

\( A \) area

\( t \) time

\( u \) velocity
\( U \)  mean axial velocity
\( \tau_\theta \)  stress tensor
\( k \)  thermal conductivity
\( \lambda \)  axial direction
\( \gamma \)  tangential or pitch-wise direction
\( \Omega \)  rotor rotational speed
\( \bar{K} \)  ideal gas constant
\( \tau \)  time period
\( n_{\text{blades}} \)  number of rotor blades
\( r \)  radius
\( r^\omega \)  vorticity-weighted radius
\( \delta^* \)  displacement thickness
\( \theta \)  momentum thickness
\( \theta^* \)  energy thickness

Subscripts
\( \text{ref} \)  reference conditions
\( 2D \)  two-dimensional geometry
\( E \)  free stream condition
Chapter 1

Introduction

An avenue to make aircraft engine compressors more compact is to decrease the number of blades in each row and the axial blade row spacing. To keep the same work input, the shaft speed or aerodynamic loading must increase - more typically, a combination of the two. The resulting highly-loaded, high Mach number (HLHM) compressors have had unexpected trends in efficiency with geometric and operational variations, particularly changes in inter-blade row spacing. This thesis examines the flow processes that lead to performance changes in HLHM axial compressors with changes in inter-blade row spacing.

HLHM compressors are characterized by a rotor relative Mach number greater than unity and shock waves that extend upstream. With decreased blade row spacing, therefore, an upstream stationary blade row encounters a stronger shock, promoting a class of unsteady interactions between shock and upstream blade row which is not present in subsonic machines.

A possible example of the result of such interactions are the changes in performance due to a decrease in axial blade row spacing at different Mach numbers, as in Figure 1.1 [1], which shows the efficiency and pressure ratio for a four stage compressor as a function of corrected mass flow at two different speeds. At lower Mach numbers, the multistage compressor has better efficiency and pressure rise as blade row...
spacing is reduced, in agreement with other low Mach number experiments [2, 3] and with computations [4, 5]. At higher Mach numbers, however, closer axial spacing between blade rows results in decreased performance. While trends observed at subsonic Mach numbers have been the subject of various studies [2-5], the flow mechanisms that lead to a reduced performance at transonic Mach numbers when blade row spacing is reduced are not well understood. This thesis identifies flow features that lead to performance variations with changes in blade row spacing.

![Figure 1.1: Effect of axial blade row spacing on performance of a 4-stage compressor [1].](image)

1.1 Previous Work

Gorrell et al. [6] examined the effect of blade row spacing on performance for an axial vane row/rotor configuration. Experiments were performed using the Air Force Research Laboratory’s ‘Stage Matching Investigation’ (SMI) rig for three different inter-
blade row axial spacings, having a mean (hub-to-tip) value of 13%, 26%, and 55% of the vane chord [6]. Choking mass flow rate, pressure ratio and efficiency all decreased as axial spacing was reduced.

Unsteady CFD simulations (using MSU Turbo [7]), conducted for the closest and farthest spacings of the SMI rig, enabled identification of a loss generating mechanism within the vane row passage [8]. Oblique shock waves that originate from the rotor intersect the upstream vane row as they sweep past, giving rise to shock waves that propagate upstream along the vane surface, as shown in Figure 1.2. In the closer blade row spacing, the shock wave becomes perpendicular to the flow direction and forms a normal shock, giving a higher entropy increase than for an oblique shock (Figure 1.2a). For the farther blade row spacing, the rotor shock is a weak compression wave at the vane trailing edge, and there is no normal shock in the upstream vane passage (Figure 1.2b).

![Comparison of wave configurations between the (a) closest and (b) farthest blade row spacing used in the SMI rig.](image)

The entropy increase, from the inlet to the vane row trailing edge plane, was 12% higher for the closest blade row spacing configuration than for the farthest spacing at the
same mass flow rate. Gorrell et al. attributed the additional entropy increase by the vane trailing edge to the normal shock in the reduced spacing, but a direct link between the two was not made. Furthermore, the entropy rise from the normal shock alone was not compared to the total shock losses or to the overall losses for the stage to determine if the normal shock was the main source of the difference in entropy generation for the different blade row spacings. The entropy generation from shock waves is assessed in the present work to determine their impact on performance as a function of blade row spacing.

In a later work by Gorrell et al. [9], CFD simulations with a more refined grid, as well as Digital Particle Image Velocimetry (DPIV) measurements, were used to examine the discrete vortices shed at the vane trailing edge that exist because of the interaction between the rotor shock and upstream stationary blade row. Within one rotor passing time, two discrete counter-rotating vortices were found to be shed at the vane trailing edge. The formation of these vortices can be explained as follows. When the rotor shock intersects the vane trailing edge, a pressure gradient is established along the surface of the blade, resulting in a flux of vorticity from the wall into the fluid, and a net circulation established around the blade. The vorticity generated on the vane is shed from the trailing edge to form a vortex. As the rotor shock moves past the vane trailing edge, the pressure gradient along the vane decreases, as does the net circulation, and a discrete vortex with opposite circulation to the previous one is shed. A vortex street which is "locked" to the rotor passing is thus formed downstream of the vane trailing edge.

From the above arguments, as the blade row spacing is reduced and there is a stronger shock impinging on the upstream blade row, there will be a larger pressure
gradient and a larger vane circulation. The length scale associated with the vortices is related to the blade thickness, and thus, a larger vane circulation results in greater vorticity within the shed vortices. The shed vortices not only contain more vorticity, but are also observed to have greater entropy as blade row spacing is reduced.

Gorrell et al. [9] developed an analytical relation for the shed vorticity as a function of vane geometry and rotor shock strength which correlated well with their computational results. The vorticity within the shed vortices is given as:

$$\omega = \left( \frac{4}{\pi \tan \phi} \right) \left( \frac{\Delta p}{\bar{p}} \right) \left( \frac{\bar{p}}{\rho_0} \right) \left( \frac{1}{\lambda} \right).$$  \hspace{1cm} (1.1)

In equation (1.1), $\phi$ is the shock angle (see Figure 1.3). $\Delta p/\bar{p}$ is the pressure rise across the shock, $\bar{p}$ is the average of the pressures ahead and behind the incoming shock, and $\lambda$ is the trailing edge blockage, i.e the ratio of trailing edge thickness to pitch. A model by Morfey and Fischer [10] was used to calculate the shock strength, $\Delta p/\bar{p}$, as a function of rotor Mach number, axial flow Mach number, and ratio of the axial distance ahead of the rotor to rotor pitch.

![Figure 1.3: Rotor shock impinging on the upstream vane row causing a net loading on the blade, and the formation of shed vortices.](image)

While the vorticity in the shed vortices was estimated, the entropy generation associated with the creation of the vortices and the additional losses as they are convected...
downstream, however, was not quantified. The loss associated with the vortices was also not compared to other loss generating mechanisms (such as shock waves) for the stage.

Zachcial and Nurnberger [11] also examined the effect of a variation in axial blade row spacing. They used two-dimensional unsteady calculations for a transonic stator/rotor combination for three different axial spacings (19.9%, 24.2% and 28.4% of the rotor chord length) operating at the same back pressure. An improvement in efficiency was found with decreased spacing, opposite to the results in Figure 1.1 and to the work by Gorrell. No discussion concerning this discrepancy was given by Zachcial and Nurnberger. The improved performance was due to a reduction in boundary layer separation within the rotor, leading to lower blade profile losses. Their observed trend in efficiency coincided with a change in the vortex trajectory within the rotor passage as blade row spacing is changed. However, the connection between a change in the vortex trajectory and the reduction in blade profile losses was inferred from the pitch-wise distribution of entropy at a location 1.5 rotor pitches downstream of the rotor, which is not as accurate as direct examination of the location where entropy is created. A numerical technique is presented in the present work to compute local entropy generation for improved assessment of flow mechanisms that impact performance.

Zachcial and Nurnberger carried out an extensive parameter study to determine the various parameters that affect the vortex pattern within the rotor, including changes in rotational speed, axial gap and stator pitch. It was found that a change in the rotational speed and the axial gap changed the relative location of the vortex within the rotor, while a change in the stator pitch has no effect. This is consistent with the work presented in this thesis, as explained in Chapter 4.
Also reported by Zachcial and Nurnberger was numerical results of the efficiency for a two-dimensional, single stage as a function of axial gap spacing, as given in Figure 1.4. They found that as axial blade row spacing increased, a minimum value of the efficiency is attained before reaching a constant value. The reason a minimum efficiency is obtained was not addressed. Based on the results of this thesis, it is postulated that the minimum can be explained by two competing effects, namely the change in vortex circulation and the relative location of vortices in the rotor with changes in spacing. A decrease in spacing can result in an increase in the entropy contained in the vortices and a decrease in efficiency, but a change in the relative location of the vortex within the rotor may give an increase in efficiency. This motivates the study of the effect of the vortex trajectory within the rotor, as its impact may lead to an optimal spacing, and is also an example of how unsteady flow events drive time-average changes that are of engineering significance.

Figure 1.4: Isentropic efficiency as a function of axial spacing for transonic operating conditions [11].
1.2 Present Work

The focus of the present work is to quantify the dominant entropy generating mechanism that lead to changes in efficiency, work and pressure rise with axial spacing between blade rows in a high Mach number, high loading compressor. The relative importance of proposed entropy generating mechanisms is assessed by computing the dissipation in the flow field. In this, the primary challenge is to accurately determine the entropy generation in regions with high spatial gradients, such as shock waves. A procedure to compute the dissipation from irreversible flow processes, as well as to isolate the entropy generation across shock waves, is developed. Applying this procedure to the vane/rotor configuration used in the studies by Gorrell [6, 8-9], it is found that the majority of the differences in entropy generation that arise from changes in blade row spacing are associated with the shed vortices. The additional entropy generation created when blade row spacing is reduced is due to higher circulation (stronger) vortices diffusing and generating additional loss as they interact with and propagate through the rotor.

A change in axial blade row spacing has two identifiable impacts on the shed vortices. First, it changes the circulation of the shed vortices (as identified by Gorrell), and second, it changes the trajectory of the vortex in the rotor passage. It is shown that for a given Mach number and geometry, the vortex trajectory within the rotor blade passage can be described by one compact, non-dimensional quantity. This non-dimensional quantity is a ratio of two time scales: the rotor period (i.e. the time for the rotor to move one rotor pitch) and the convective time for the shed vortices to travel the length of the axial gap. These determine the relative location of the vortices that enter the
rotor passage. The vortex trajectory, in turn, can impact the rotor performance as vortices change the rotor flow field and interact with the rotor boundary layers.

To determine the effect of vortex trajectory on rotor efficiency, two-dimensional computations were conducted for two different axial blade row spacings. The two blade row spacings were chosen such that the strength of the rotor shock at the vane row trailing edge was the same (in order to maintain the same vortex strength). Because of the trajectory change, however, one configuration had two vortices in the rotor passage core flow (i.e. outside the boundary layer), while the other had one vortex in the core flow and one located near the blade surface, mainly within the boundary layer. The largest difference in entropy generation between the two configurations occurs downstream of the rotor, due to the mixing of the different flow fields at the rotor trailing edge plane.

1.3 Technical Objectives

In this thesis, mechanisms that impact performance of a transonic rotor due to changes in upstream blade row spacing are determined. For this, a computational method is developed to quantify local entropy generation. Isolating the entropy generation associated with different flow features, including shock waves, was one objective of this work. A second objective is to determine the impact of vortex trajectory on stage efficiency, work input and pressure rise.

The research questions to be answered are:

- How do the dominant entropy generation mechanisms vary with axial blade row spacing?
- What is the performance impact of a change in the trajectory within the rotor of vortices from the upstream vane row?
1.4 Thesis Scope and Content

To answer these research questions, a detailed interrogation of numerical computations has been carried out. Chapter 2 describes the CFD model, the geometry and the performance metrics for the computational simulations. Chapter 3 defines a numerical technique to isolate the regions of entropy generation. In Chapter 4, a non-dimensional parameter is defined that describes the relative location of vortices in the rotor, and the performance changes associated with changes in this parameter are assessed using two-dimensional, unsteady computations. Chapter 5 summarizes the findings and conclusions, and Chapter 6 gives suggestions for future work.

1.5 Research Contributions

- A framework and computational methodology is developed for quantifying local entropy generation in transonic compressors, even in regions with high spatial gradients, such as shock waves.
- The impact of vortex trajectory within the rotor passage on rotor efficiency, pressure rise and work input is determined. A non-dimensional parameter to characterize the trajectory of the vortices is defined.
Chapter 2

Numerical Simulations

Numerical simulations were conducted to answer the research questions outlined in the previous chapter. To determine the entropy generating mechanisms responsible for the change in compressor performance with axial spacing, three-dimensional calculations were performed using the geometry and code employed by Gorrell [6, 8-9]. The effect of a change in the vortex trajectory within the rotor was studied via two-dimensional calculations. The present chapter describes the CFD code, the geometries, and the performance metrics for the numerical computations.

2.1 Numerical Code

Numerical simulations were conducted using MSU Turbo Version 4.1 [7], an unsteady, three-dimensional, viscous code that solves the Reynolds Averaged Navier-Stokes (RANS) equations. The equations are solved in the reference frame of each blade row. The code employs a finite volume solver, with a $\kappa-\varepsilon$ turbulence model. Communication between blade rows in their respective reference frames occurs across a sliding plane that interpolates information from one blade row to the other.

To reduce computer time and memory required to perform the numerical experiments, temporal phase lag boundary conditions were used [12-15]. Temporal phase lag boundary conditions rely on the assumption that the flow field associated with
two interacting blade rows has a temporal periodicity related to the blade count of the
blade rows. More specifically, the flow field within the blade passage will repeat itself
every time the relative position of the adjacent blade row is the same. The phase lag
approximation permits the replacement of a full wheel or spatially periodic computation
by a single blade passage within each blade row. The current geometry has 24 stator and
33 rotor blades. If periodic boundary conditions were used, a sector with 8 stator blades
and 11 rotor blades would be required, implying much more computational time and
computer memory than simulations with the temporal phase lag approximation. Using
the phase lag approximation, a full wheel can be constructed at any instant in time using
the computed flow field of the individual passage from previous instants in time. Details
of the full wheel reconstruction are given by Wang and Chen [15].

An assumption associated with the use of phase lag boundary conditions is that
the lowest frequency of any important unsteady phenomenon is the blade-passing
frequency of the adjacent blade row. For example, vortex shedding, rotating stall, or flow
separation that is at a lower frequency than the blade passing frequency would not be
captured. Use of the phase lag boundary condition is an appropriate approximation for
unsteady blade row interactions where the frequency of unsteadiness in one blade row is
dominated by the adjacent blade row passing frequency. The results from phase lag
computations and experiment have been compared in a previous study [9], and the phase
lag approximation captured the flow features that arise from the blade row interactions,
including the vortex shedding from the inlet vanes due to the impingement of the rotor
shock on the vane.
Uniform stagnation pressure and temperature are specified at the inlet of the computational domain. For three-dimensional simulations, at the exit of the computational domain, the static pressure at the hub is specified, and simple radial equilibrium is used to specify the radial distribution of static pressure. For the two-dimensional simulations, the static pressure is specified at the exit. The convergence criteria was that the time-averaged mass flow rate at the vane row inlet and at the exit of the domain were within 0.1% of each other, and that the efficiency and mass flow were periodic with blade passing frequency.

Turbo solves the flow field using primitive variables at cell centers. The Turbo output data is therefore chosen to be the values at the cell center (instead of using the conventional Plot3D output, which interpolates to cell nodes). Use of cell-center output means that information is not lost in the interpolation scheme associated with Plot3D. Visualization of the output was accomplished with the codes developed by Villanueva [16].

2.1.1 Time Discretization and Time-averaging

The number of time steps for each blade period was chosen based on the desired temporal resolution of the blade row interactions. The frequency of unsteadiness within one blade row is determined by the blade passing frequency of the adjacent blade row, and resolving twenty harmonics of the blade passing frequency should well capture the flow features deriving from unsteady blade row interactions, including vortex shedding. According to Nyquist’s theorem, a harmonic must be sampled at least twice per cycle in order to be resolved, so 40 samples were taken for one blade-passing period. In the vane/rotor configuration studied here, the rotor blade passing frequency in the vane
reference frame is higher than the vane blade passing frequency in the rotor reference frame. The vane blade row simulation employed a time increment equal to \(\frac{1}{40}\)th of the rotor blade passing period, so the vane blade row is resolved in 40 time steps and the rotor blade row is resolved in 55 time steps (based on the vane to rotor blade ratio of 8:11).

Within the Turbo code, specifying a small number of time steps requires more iteration at each time step to reach convergence; more time steps means less iteration. The number of time steps chosen was based on experience gained in the simulations conducted by Gorrell [17], and is given in Table 2.1. By choosing a larger number of time steps, the solution of the flow field captures more harmonics than required, achieving greater accuracy.

Instead of time-averaging over one blade period using the computed results at every time instant (based on the number of time steps per period in Table 2.1), the unsteady flow field signature is captured up to the twentieth harmonic of the blade passing frequency. The vane blade row is time-averaged over 40 time instants and the rotor over 55 time instants, both at increments of four time steps apart. In the two-dimensional computations, the vane to rotor blade ratio is simplified to 2:3 (as will be discussed in section 2.3), and the rotor is averaged over a total of 60 time instants.

<table>
<thead>
<tr>
<th>Blade row</th>
<th>3D</th>
<th>2D</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vane</td>
<td>220</td>
<td>240</td>
</tr>
<tr>
<td>Rotor</td>
<td>160</td>
<td>160</td>
</tr>
</tbody>
</table>

Table 2.1: Number of time steps per period used in Turbo code.
2.2 Three-dimensional Calculations

2.2.1 Geometry

The three-dimensional geometry is based on the Air Force Research Laboratory ‘Stage Matching Investigation’ (SMI) rig, which is designed to study changes in performance with variations in axial blade row spacing for a highly-loaded, high Mach number compressor. The rig was composed of three blade rows: a wake generator or inlet guide vanes (IGV), a rotor and a stator. It was also run as a vane and rotor-only combination [6, 8-9]. The latter configuration is studied here for two axial blade row spacings, denoted as “close” and “far”, and given in Table 2.2.

Table 2.2: Axial blade row spacing $L$ for close and far configurations, normalized by the axial rotor chord length $c_r$.

<table>
<thead>
<tr>
<th>Spacing</th>
<th>$L/c_r$ (mean)</th>
<th>$L/c_r$ (hub)</th>
<th>$L/c_r$ (tip)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Close</td>
<td>0.22</td>
<td>0.13</td>
<td>0.27</td>
</tr>
<tr>
<td>Far</td>
<td>0.86</td>
<td>0.80</td>
<td>0.95</td>
</tr>
</tbody>
</table>

The straight vane blades are designed to emulate the typical loss profile from a front stage stator in a high speed axial compressor. Details of the vane design are given by Chriss et al. [18]. The aerodynamic design parameters for the rotor are given in Table 2.3. The rotor tip clearance is 0.6% of the chord. The ratio between the number of vanes and rotor blades is 8:11.
Table 2.3: SMI Aerodynamic Design Parameters [6].

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Rotor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of blades</td>
<td>33</td>
</tr>
<tr>
<td>Aspect Ratio (average)</td>
<td>0.961</td>
</tr>
<tr>
<td>Inlet Hub/Tip Ratio</td>
<td>0.750</td>
</tr>
<tr>
<td>Tip speed, corrected m/s</td>
<td>341.37</td>
</tr>
<tr>
<td>RPM</td>
<td>13509.0</td>
</tr>
<tr>
<td>$M_{rel} \text{ LE} M_{rel} \text{ Hub}$</td>
<td>0.963</td>
</tr>
<tr>
<td>$M_{rel} \text{ LE Tip}$</td>
<td>1.191</td>
</tr>
<tr>
<td>LE Tip Dia., m</td>
<td>0.4825</td>
</tr>
</tbody>
</table>

The grid was created using the Average Passage Grid (APG) generator of Beach [19]. Gorrell et al. have determined that the grid provides sufficient resolution to capture the vortex shedding by comparing the numerical results to DPIV measurements [9], and their grid was used for the calculations described here. The number of grid points for both the vane and rotor are given in Table 2.4. Additional details of the grid are provided by Turner et al. [20].

Table 2.4: Number of grid points in axial, radial and pitch-wise directions for the three-dimensional calculations.

<table>
<thead>
<tr>
<th>Blade row</th>
<th>Axial</th>
<th>Radial</th>
<th>Pitch-wise</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vane close</td>
<td>138</td>
<td>71</td>
<td>61</td>
</tr>
<tr>
<td>Vane far</td>
<td>230</td>
<td>71</td>
<td>61</td>
</tr>
<tr>
<td>Rotor</td>
<td>189</td>
<td>71</td>
<td>81</td>
</tr>
</tbody>
</table>

2.2.2 Performance Metrics

The experimentally measured performance metrics were obtained by Gorrell for the far and close configurations. The pressure ratio and efficiency are consistently lower for the close spacing than for the far spacing, as in Figure 2.1. The pressure ratio and efficiency were based on measurements at the inlet and at an exit plane 0.9 axial rotor chords downstream of the rotor trailing edge.
The efficiency, mass-averaged total pressure ratio, and mass-averaged total temperature rise, obtained from the CFD simulations, for the two configurations at the same corrected mass flow, are given in Table 2.5. The adiabatic efficiency is calculated as:

$$
\eta = \frac{TR \left( \exp \frac{\Delta s}{R} \right)^{\frac{\gamma-1}{\gamma}}}{TR - 1},
$$

(2.1)

where the stagnation temperature ratio $TR$ and entropy rise $\Delta s$ are mass- and time-averaged over the measurement plane. The stagnation temperature rise, stagnation pressure ratio, and efficiency are lower for the close spacing configuration than for the far
spacing configuration. The two configurations described here are further compared in Chapter 3 to determine where the differences in entropy generation arise.

Table 2.5: Mass flow rate, mass-averaged total temperature rise, mass-averaged total pressure ratio, and adiabatic efficiency from three-dimensional calculations for far and close spacing configurations. Exit values are time-averaged on a plane 0.9 axial rotor chords downstream of the rotor trailing edge.

<table>
<thead>
<tr>
<th>Spacing</th>
<th>Mass flow (kg/s)</th>
<th>Mass-averaged $T_t$ rise</th>
<th>Mass-averaged $P_t$ ratio</th>
<th>Efficiency $\eta$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Close</td>
<td>14.89</td>
<td>0.221</td>
<td>1.846</td>
<td>0.891</td>
</tr>
<tr>
<td>Far</td>
<td>14.89</td>
<td>0.219</td>
<td>1.872</td>
<td>0.897</td>
</tr>
</tbody>
</table>

2.3 Two-dimensional Calculations

2.3.1 Geometry

A two-dimensional grid was generated using the blade geometry from the far spacing three-dimensional grid at 65% span. This radial span represented the average stagger angle of the 3D grid from hub to tip. The same code used for the 3D calculations was used for the 2D calculations. The geometry was created by placing two radial planes close together and at a large radius, $R$, so that $\Delta r/R$ was 0.17% (in the simulation, the radius $R$ was made 110 times the original radius). The vane to rotor blade ratio was also changed, for simplicity, to 2:3. The configuration created using an axial spacing equal to the spacing at 65% span is referred to as far$_{2D}$. The configuration with the axial spacing reduced by 11% from far$_{2D}$ will be referred to as close$_{2D}$. The axial spacing is 0.81 and 0.91 of the axial rotor chord length for close$_{2D}$ and far$_{2D}$, respectively.

The effect of a change in the shed vortex trajectory on performance was examined using the close$_{2D}$ and far$_{2D}$ configurations. Chapter 4 describes how a change in axial spacing can affect the trajectory of the wake vortices through the rotor passage. The
spacing change was chosen such that the rotor shock strength at the vane trailing edge plane (which set the strength of the shed vortices) differed by less than 2%. The difference in Mach number at the vane trailing edge plane between the two configurations was 0.01 (the Mach number is 1.05 in close$_{2D}$ and 1.06 in far$_{2D}$).

The number of grid points for both configurations is given in Table 2.6. The same number of grid points was maintained for the close$_{2D}$ and far$_{2D}$ vanes, but the distance between grid points in the axial direction was decreased for close$_{2D}$. Since the mesh in the far$_{2D}$ was already sized to capture the important flow features in the axial gap region, the finer mesh in close$_{2D}$ will also capture the important flow features.

Table 2.6: Number of grid points in axial, radial and pitch-wise directions for the two-dimensional calculations.

<table>
<thead>
<tr>
<th>Blade row</th>
<th>Axial</th>
<th>Radial</th>
<th>Pitch-wise</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotor</td>
<td>189</td>
<td>2</td>
<td>81</td>
</tr>
<tr>
<td>Vane close$_{2D}$</td>
<td>230</td>
<td>2</td>
<td>61</td>
</tr>
<tr>
<td>Vane far$_{2D}$</td>
<td>230</td>
<td>2</td>
<td>61</td>
</tr>
</tbody>
</table>

2.3.2 Performance Metrics

A comparison of performance between far$_{2D}$ and close$_{2D}$ was made at the same mass flow. The stagnation pressure ratio and stagnation temperature rise at the mixed-out conditions are given in Table 2.7, while the efficiencies at the rotor exit and far downstream are given in Table 2.8. All are lower for close$_{2D}$ compared to far$_{2D}$. Close$_{2D}$ exhibits 3% less work than far$_{2D}$ and has a 1.7% lower stagnation pressure ratio. The difference in efficiency at the rotor exit is 0.01 points (lower in close$_{2D}$ than far$_{2D}$), which is not of interest, but at the far downstream mixed-out state, the difference in efficiency is greater, with close$_{2D}$ 0.3 points lower than far$_{2D}$. The majority of the difference in
entropy generation thus occurs downstream of the rotor trailing edge, as is discussed in Chapter 4.

Table 2.7: Mass flow rate, mass-averaged total temperature rise, and mass-averaged total pressure from two-dimensional calculations for far2D and close2D configurations. Values are time-averaged at downstream infinity.

<table>
<thead>
<tr>
<th>Spacing</th>
<th>Mass flow (kg/s)</th>
<th>Mass-average Tt rise</th>
<th>Mass-average p/r ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Close2D</td>
<td>1148.47</td>
<td>0.187</td>
<td>1.739</td>
</tr>
<tr>
<td>Far2D</td>
<td>1149.54</td>
<td>0.193</td>
<td>1.772</td>
</tr>
</tbody>
</table>

Table 2.8: Efficiency η for far2D and close2D spacing configurations at the rotor exit and at downstream infinity.

<table>
<thead>
<tr>
<th>Spacing</th>
<th>Rotor Exit</th>
<th>Downstream infinity</th>
</tr>
</thead>
<tbody>
<tr>
<td>Close2D</td>
<td>0.9405</td>
<td>0.9205</td>
</tr>
<tr>
<td>Far2D</td>
<td>0.9406</td>
<td>0.9235</td>
</tr>
<tr>
<td>Difference</td>
<td>0.0001</td>
<td>0.0030</td>
</tr>
</tbody>
</table>
Chapter 3

Quantification of Entropy Sources

The differences between the performance metrics for the “close” and “far” configurations, as defined from three-dimensional computations, were described in Chapter 2. Gorrell et al. [8, 9] described two possible mechanisms to explain the lower performance in the close spacing configuration compared to the far spacing. The first was the oblique shock increasing in angle (towards the direction perpendicular to the free stream) inside the vane row [8]. The second was the entropy associated with the shed vortices as they are formed at the vane trailing edge when the blade row spacing is reduced [9]. However, the entropy generation associated with these two mechanisms was not compared quantitatively with the overall losses for the stage. In this chapter, the dominant source of entropy generation that leads to the performance differences between the far and close spacing is identified and quantified, using a new numerical technique to calculate the local dissipation. In addition, a method to isolate entropy generation within shock waves is also developed.
3.1 Sources of Entropy Generation

To demonstrate where the entropy generation occurs in the two three-dimensional configurations, the entropy flux at different axial locations is presented in Figure 3.1. The close spacing has a higher entropy increase than the far spacing within the rotor and downstream of the rotor, leading to a lower efficiency for the close spacing. The higher overall entropy rise throughout the domain for the close spacing occurs despite the fact that the entropy increase from the inlet of the domain to the rotor leading edge is lower in the close spacing than in the far spacing configuration.

![Figure 3.1: Entropy flux as a function of axial location for far and close spacing. Entropy values referenced to $p_{ref} = 1$ atm, $T_{ref} = 300$K.](image)

The conclusion from Figure 3.1 is that the two entropy-generating mechanisms just described, which occur upstream of the rotor, do not account for the reduced

---

1 In the figure, it should be noted that the physical distance between the vane trailing edge and the rotor leading edge is different for the far and close spacings.
performance in the close spacing. These two mechanisms imply that the additional entropy generation in the close spacing occurs upstream of the rotor leading edge plane. If so, the entropy flux into the rotor would have to be higher into the rotor in the close spacing. This is not the case.

To explain the larger entropy increase downstream of the rotor leading edge plane in the close configuration, the rotor shock and the shed vortices are examined. Losses associated with the shed vortices as they convect downstream have not previously been assessed. Figure 3.2 shows entropy contours from numerical simulations for far and close spacings at the mid-span radius. Entropy is non-dimensionalized as $T\Delta s/(\text{in} \rho A)^{\frac{3}{2}}$ and referenced to $p_{\text{ref}} = 1$ atm, $T_{\text{ref}} = 300$K. In the close spacing, higher entropy is observed in the vortices at their inception than in the far spacing, as identified by Gorrell. However, as these vortices move downstream, they diffuse and interact with the rotor boundary layer and wake, generating more entropy. An issue to be addressed is whether the entropy generation associated with the shed vortices as they move through the blade passage is a major contributor to the total difference in entropy generation between the far and close spacing configurations.
Figure 3.2: Entropy contours at mid-span for (a) far spacing, (b) close spacing. All plots are of the same scale. The lower contour plots correspond to the boxed regions in the upper plots, and have 20 evenly-spaced intervals.

Rotor shock waves are another source of entropy generation. The shock wave location can be highlighted using the velocity divergence field. The continuity equation is:

$$\nabla \cdot \bar{u} = -\frac{1}{\rho} \frac{D\rho}{Dt}.$$  \hspace{1cm} (3.1)

Regions of compression have a negative value of the divergence of velocity, and expansion regions have a positive value. Figure 3.3 is a contour plot of the divergence of velocity at the mid-span for both configurations. The divergence of velocity is non-dimensionalized by the tip speed (which is equal to $\sqrt{\gamma R T_{ref}}$) and the axial rotor chord length. The shock structure is different for the two spacings, and is more unsteady with
the close spacing compared to the far spacing. The entropy generation within the rotor due to shock waves is quantified later in this chapter.

Figure 3.3: Divergence of velocity contours at mid-span for (a) far spacing, (b) close spacing. Blue regions describe regions of compression, and red describe regions of expansion.

3.2 Numerical Approach to Compute Dissipation

In this section, the numerical implementation of the equations to calculate the local entropy generation will be described. The entropy increase within a specified control volume is then quantified to determine where differences exist between the close and far spacing configurations.

The rate of entropy change of a fluid particle is:

$$\frac{\rho Ds}{Dt} = \frac{1}{T} \tau_{ij} \frac{\partial u_i}{\partial x_j} + \frac{1}{T} \frac{\partial}{\partial x_i} \left( k \frac{\partial T}{\partial x_i} \right).$$  \hspace{1cm} (3.2)

The two terms on the right-hand side of equation (3.2) represents entropy changes due to viscous effects and heat transfer, respectively. The former is irreversible, but the latter includes both reversible and irreversible changes. All irreversible processes in this thesis will be referred to as dissipative processes, where entropy generation is equivalent to lost
work. The "physical dissipation" will be defined as the entropy generation due to the action of viscosity and irreversible heat transfer, calculated from the gradients in velocity and temperature. For a control volume encompassing the rotor shock waves, as in Figure 3.4, reversible heat transfers across the control surface are negligible compared to entropy generation within the control volume. Therefore, the physical dissipation should (theoretically) capture the entropy generation across a shock wave.

![Figure 3.4: Dashed line represents a control volume around the rotor shock wave.](image)

However, a problem arises when the physical dissipation is computed from gradients in velocity and temperature in the region of the shock wave, because the grid is not dense enough to accurately compute the spatial derivatives. Therefore, instead of computing the entropy generation from gradients in velocity and temperature in the right-hand side of equation (3.2), the local entropy rise can be computed from the left-hand side of equation (3.2) directly, which will be referred to as the "computational dissipation," i.e.

\[
\frac{\partial(\rho s)}{\partial t} + \nabla \cdot (\rho \vec{u} s).
\]  

(3.3)

The first term in equation (3.3) represents the time-rate of change of entropy per unit volume. In the code, it is calculated using a centered-difference scheme in time. The
second term represents the net entropy flux for each computational cell. This is computed by interpolating fluid properties to each cell boundary to define the net entropy flux out of the cell volume at every instant in time.

The computational dissipation allows adequate evaluation of entropy generation in regions with shock waves because the entropy is calculated from primitive variables, which are obtained by solving the conservation equations in the Turbo simulation. In the Turbo code, the numerical scheme includes additional dissipation, other than the physical dissipation to accurately calculate entropy generation. This additional dissipation is referred to as "numerical dissipation." The total dissipation (or entropy generation) is therefore equal to the physical dissipation (computed from the gradients in velocity and temperature) plus the numerical dissipation (implemented by the code). As the grid mesh size goes to zero, the numerical dissipation also vanishes.

Using the new definitions for the calculated dissipation, the computational dissipation is equal to the sum of the physical dissipation, numerical dissipation, and the entropy changes due to reversible heat transfers across the control surface. As stated previously, these reversible heat transfers are negligible in the region of shock waves. They are identically zero if the bounding surface of the control volume of interest is adiabatic.

The computational dissipation, when time-averaged and integrated over a specified control volume, is equal to the time-averaged net entropy flux out of that control volume. This is because the first term in equation (3.3) vanishes due to periodicity of the flow field, i.e.:

\[
\frac{1}{\tau} \int \int \left( \frac{\partial (\rho s)}{\partial t} + \nabla \cdot (\rho \bar{u} s) \right) dt dV = \frac{1}{\tau} \int \int (\rho \bar{u} s) \cdot \bar{n} dA dt.
\] (3.4)
When the computational dissipation is integrated over a control volume with an adiabatic surface, the computational dissipation should equal the entropy generation for that volume.

Where heat transfer terms are negligible, the computational dissipation should always be positive. Negative values, however, arise because the computational dissipation is calculated using a centered-difference scheme as opposed to the up-winding scheme used by Turbo. Entropy-generating flow features (e.g. shock waves, vortices) are associated with both positive and negative values of the computational dissipation that are located close together in the region of the flow feature. When the computational dissipation is integrated over a control volume that encompasses both the positive and negative values of the computational dissipation associated with these flow features, the entropy generation is accurately captured.

The contribution of the physical dissipation and the computational dissipation can be assessed independently in regions that have both shock waves and shear layers. This allows us to determine how well the physical dissipation represents the actual entropy rise for a control volume. Figure 3.5 shows the entropy rise within a control volume defined from the vane trailing edge to a specified axial location and from hub to tip (these are adiabatic surfaces). Within the first 40% of the axial gap, which is the region in which the shed vortices are formed, the computational and physical dissipation are in good agreement, implying that the physical dissipation accurately captures the entropy generation associated with shear layers. After this point, the two dissipation schemes diverge, and the physical dissipation under-estimates the computational (i.e. actual) entropy flux. The divergence occurs because closer to the rotor, the shock wave is
stronger and the physical dissipation is unable to resolve the spatial gradients associated with the shock wave. The inability of the physical dissipation to capture the shock entropy rise demonstrates the necessity to use the computational dissipation as the proper measure of entropy creation.

![Graph](image)

**Figure 3.5**: Comparison of entropy rise calculated from the computational and physical dissipation in axial gap for far spacing. Entropy values are calculated with respect to the vane trailing edge. Solid lines are calculated from the computational dissipation, and dashed lines from the physical dissipation.

### 3.3 Procedure to Calculate Shock Entropy Rise

In Section 3.1, differences in the shock behavior were observed for the different configurations. The corresponding difference in entropy generation from the shocks will now be quantified. To isolate the shock waves computationally, the divergence of velocity is used as a marker and the shock waves are highlighted by marking regions where the divergence of velocity is below a specified threshold value.
Figure 3.6 illustrates the procedure to isolate the entropy rise associated with shock waves. In Figure 3.6a, two contour plots of the divergence of velocity are shown at the mid-span in the far spacing. On the left, the divergence of velocity is given everywhere in the flow field, and on the right, the divergence of velocity only in the shock region is plotted. Negative values of the divergence of velocity define compression regions, and the shocks (the dark blue regions) can be identified ahead of the rotor. To calculate the entropy rise across the shock wave, a control volume is placed around the shock wave. The geometry of the control volume around the shock wave is
determined by specifying a threshold value for the divergence of velocity. If the divergence of velocity is below the threshold value at any cell, that cell is specified as part of the shock wave control volume.

Implementing the above procedure of identifying the shock region is referred to as "masking", because the shock waves are essentially masked from the rest of the flow field. In Figure 3.6b, positive values of the computational dissipation are plotted in the contour plot on the left. Negative values were not plotted because they would reduce the clarity of the figure, but are immediately adjacent to regions where the computational dissipation is positive. Negative values are important when integrating the computational dissipation to find the entropy rise associated with the shock wave, as stated in Section 3.2. The computational dissipation is non-dimensionalized as $T(\text{comp. dissip.}) c_s / \rho (m / \rho A)^3$, where \textit{comp. dissip.} represents the quantity in equation (3.3). By applying the mask of the divergence of velocity, the computational dissipation in the shock region is isolated, as seen in the contour plot on the right of Figure 3.6b. Integrating the "masked" computational dissipation in space and time gives the entropy rise associated with the rotor shock.

![Figure 3.7: Schematic of the profile of divergence of velocity in shock region. A primary mask isolates the shock wave, and a secondary mask ensures the entire shock region is captured.](image-url)
One issue in this procedure concerns setting the magnitude of the threshold. The shock wave encompasses a finite region within which the divergence of velocity can take on a range of values below the threshold value, as depicted in Figure 3.7. The choice of threshold value is critical. If it is too high, it can include cells associated with flow features unrelated to the shock. Therefore, a primary threshold value (called the primary mask) is used to identify the general location of the shock, and then a secondary threshold value (called the secondary mask) is applied to the immediate surroundings to ensure capture of the entire shock region. If the divergence of velocity is below the secondary threshold in the cells that neighbor the region where the primary mask is valid, those cells are also considered to be in the shock region. This ensures that the complete shock structure, and only the shock structure, is included in the mask and reduces numerical error when the primary mask does not capture the entire shock region.

To apply the above procedure, three parameters must be specified: the primary and secondary threshold values, and the number of neighboring cells around the cell where the primary mask is valid to evaluate the secondary mask value. The values for all three parameters, chosen to capture the entire shock region at all spans, were varied to test sensitivity of the results. It was found that the value of each of the three parameters does not change the conclusion that the majority of the differences in entropy generation between configurations is not from shock waves. Table 3.1 shows the results from four values of the primary mask that were used to compare the contribution from inside and outside the masked region to the overall difference in rotor entropy rise between the far and close configurations. To encompass the entire shock region for all values of the
primary mask in Table 3.1, the secondary mask was chosen to be -0.2, and was evaluated within a range of two cells around the primary mask.

Table 3.1: The difference in entropy generation within the rotor for the far and close configurations is divided into the differences inside and outside the masked (shock) region. The difference is measured for a range of primary mask values of the divergence of velocity.

<table>
<thead>
<tr>
<th>$\nabla \cdot \vec{v}$ - primary mask</th>
<th>Mask</th>
<th>Outside Mask</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>27%</td>
<td>73%</td>
</tr>
<tr>
<td>-1</td>
<td>24%</td>
<td>76%</td>
</tr>
<tr>
<td>-1.5</td>
<td>14%</td>
<td>86%</td>
</tr>
<tr>
<td>-2</td>
<td>2%</td>
<td>98%</td>
</tr>
</tbody>
</table>

Over the range of primary mask values in Table 3.1, the close spacing has a higher entropy rise both inside and outside the masked region. The maximum difference in entropy rise between the far and close spacing within the masked region is 27% of the total difference. The rest of the difference in entropy increase is outside the shock region. Lowering the magnitude of the primary threshold reduces the computed contribution of the shock entropy rise to the total entropy rise, but the conclusion that the majority of the losses occur outside the shock region is unchanged, even with the magnitude of the primary mask set to zero. This conclusion holds when the end wall regions are removed from the control volume over which the computational dissipation is integrated, and in fact, the shock losses in the far spacing are measured to be higher than in the close spacing when the non-dimensionalized divergence of velocity is -2, as seen in Table 3.2.

Table 3.2: Results from Table 3.1, excluding the end wall region. A positive percentage value indicates that the close configuration has a higher entropy generation than the far configuration.

<table>
<thead>
<tr>
<th>$\nabla \cdot \vec{v}$ - primary mask</th>
<th>Mask</th>
<th>Outside Mask</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>37%</td>
<td>63%</td>
</tr>
<tr>
<td>-1</td>
<td>23%</td>
<td>77%</td>
</tr>
<tr>
<td>-1.5</td>
<td>9%</td>
<td>91%</td>
</tr>
<tr>
<td>-2</td>
<td>-9%</td>
<td>109%</td>
</tr>
</tbody>
</table>
The conclusion is that the entropy generation due to shock waves is not the dominant mechanism responsible for the performance difference observed with changes in spacing. The main difference between the far and close spacing configurations must therefore be associated with differences in entropy generation from the vortices within the rotor blade passage. The next chapter focuses on these shed vortices and their effect on the overall stage performance as they propagate through, and interact with, the rotor.
Chapter 4

Vortex Trajectory within the Rotor

A change in axial inter-blade row spacing has two identifiable effects on the shed vortices from the upstream vane row: 1) a change in the strength of the vortices, and 2) an alteration in vortex trajectory within the rotor. The former is the increased circulation of the shed vortices as axial blade row spacing is decreased, as described by Gorrell [9]. The second effect, the change in the relative location of the vortices in the rotor blade passage, is depicted in Figure 4.1, which conceptually shows vortex trajectories for two axial spacings. This chapter focuses on the parameters that affect the vortex trajectory, and how the changes in trajectory affect stage performance.

![Figure 4.1](image)

4.1 Parameters that Impact Vortex Trajectory within the Rotor

Shed vortices from the upstream vane trailing edge move downstream across the axial blade row gap and then through the rotor blade passage. A key observation is that
the pitch-wise location of the vortices within the rotor passage is defined by the pitch-wise location of the vortex when it crosses the rotor leading edge plane. For a given Mach number and blade geometry in this two-dimensional flow, the pitch-wise location of the vortices is a function of the ratio between two times scales: the time it takes for a vortex to travel the length of the vane-rotor gap, and one rotor period (i.e. the time for the rotor to move one rotor pitch). To elaborate, let the time origin, \( t=0 \), be the instant the vortex is shed. At this instant, the rotor is at a certain position with respect to the upstream vane row. As the vortex travels the gap, the rotor moves from its position at \( t=0 \). Therefore, the relative location of the vortex in the rotor thus depends on how far the rotor has rotated in the time it takes for the vortex to travel the length of the gap.

Based on the above arguments, there are two time scales of interest. One is the convective time for a vortex to travel the length of the gap \( (\tau) \), i.e.

\[
\tau = \frac{L}{U}
\]

(4.1)

where \( L \) is the gap length, and \( U \) is the mean axial velocity. The second is the rotor period, which is the time for the rotor to move one rotor pitch \( (\tau') \):

\[
\tau' = \frac{2\pi / n_{\text{blades}}}{\Omega}
\]

(4.2)

where \( n_{\text{blades}} \) is the number of rotor blades, and \( \Omega \) is the rotor rotational speed. The vortex trajectory within the rotor blade passage depends on the ratio of \( \tau'/\tau \), except in the cases where the ratios are integer multiples. An integer multiple of the time ratio (or similarly, a multiple of the rotor period \( \tau' \)) is equivalent to the rotor moving through that same multiple value of the rotor pitch, and the vortex trajectory is unchanged.
It should be emphasized that the vortex shedding from the upstream vane row is directly linked to the rotor blade passing frequency by the rotor pressure field. More specifically, the kinematics of the rotor shock impinging on the upstream vane row dictate that the vortex shedding from the vane trailing edge occurs at a frequency equal to the rotor blade passing frequency. The consequence of this relationship is that the rotor is at the same relative position to the upstream vane row when vortices of the same sign are shed.

The ratio of the two time scales, the convective time scale and the rotor period, defines a dimensionless parameter $B3$:

$$B3 = \frac{L\Omega}{U} \left( \frac{1}{2\pi/nblades} \right)$$  \hspace{1cm} (4.3)

For a vortex shed from the vane trailing edge in a two-dimensional flow, a change in the parameter $B3$ (in particular the axial gap spacing $L$, which is the present variable of interest) results in a change in the vortex trajectory through the rotor.

The above discussion pertains to a two-dimensional flow. For a three-dimensional flow, a change in blade row spacing will result in a change in the vortex trajectory at every radial plane. At each radial span in an axial compressor, the vortex trajectory may be beneficial or detrimental with regards to performance. Altering the vortex trajectory at each radial span could lead to an enhanced stage performance. For example, the vortex trajectory within the rotor can be tuned at each radial span by changing the $B3$ parameter or the stagger angle of the blade.
4.2 Vortex Trajectories in the Two-dimensional Computations

To assess the effect of a change in the vortex trajectory on rotor performance, two-dimensional calculations have been conducted, as outlined in chapter two. The configurations tested, referred to as "close\textsubscript{2D}" and "far\textsubscript{2D}," had axial blade row spacings of 0.81 and 0.91 of the chord length, respectively. The value of $B3$ for close\textsubscript{2D} is 16.7 and for far\textsubscript{2D} is 18.8. At the same vane inlet corrected mass flow, close\textsubscript{2D} experiences a 3% decrease in work input and a 1.7% lower stagnation pressure ratio compared to far\textsubscript{2D}. The efficiency at the rotor trailing edge is 0.01 points lower in close\textsubscript{2D} compared to far\textsubscript{2D}, however, close\textsubscript{2D} is 0.3 points lower far downstream.

These changes can be linked to the different vortex trajectories within the rotor passages for the two configurations, as pictured in the entropy contours of Figure 4.2. For far\textsubscript{2D}, a clockwise vortex enters the mid-passage of the rotor, while a counterclockwise vortex intersects the rotor leading edge and remains near the blade. For close\textsubscript{2D}, both vortices move through the rotor but are away from the blades.
Time-averaged entropy contours in the rotor frame of reference are given in Figure 4.3. The largest values of the time-averaged entropy are along the blade surfaces, representing the rotor boundary layers. Away from the blades, streaks of smaller time-averaged entropy than that in the boundary layers delineate the path of the vortices, made clearer by dashed lines. A single streak of entropy is visible in far_{2D}, while two streaks are seen at a different relative location to the blades in close_{2D}. 

Figure 4.2: Entropy contours for (a) far_{2D} and (b) close_{2D}. Comparison of both configurations shows different relative locations of vortices in the rotor blade passages. All plots are of the same scale. The lower contour plots correspond to the boxed regions in the upper plots, and have 20 evenly-spaced intervals.
Figure 4.3: Time-averaged entropy contours within the rotor passage for (a) far2D and (b) close2D. Streaks of entropy within the passage delineate the path taken by the vortices. Two streaks are visible in close2D, while only one is seen in far2D because the other vortex is located within the rotor boundary layer.

4.3 Vortex Strength and Size in Gap Region

In this section, the performance differences measured between the two configurations will be shown to be due to changes in the vortex trajectory, and not to changes in vortex strength or size that occur with spacing. The term vortex strength refers to the net circulation, and vortex size refers to the vortex physical extent (such as the equivalent radius). The vortex size at the rotor leading edge plane varies with spacing because of the different lengths over which the vortices can diffuse before they enter the rotor. If the vortex strength or size is different at the rotor leading edge plane in the two configurations, there can be differences in entropy generation processes not associated with a change in vortex trajectory. These entropy generation processes include different levels of viscous dissipation, and different interactions with the rotor shock and boundary.
layer (e.g. a vortex of larger circulation and size can influence the velocity field more strongly). It is shown below that differences in vortex circulation and size are negligible between the two configurations.

The difference in vortex strength is first assessed by calculating the vortex circulation for clockwise rotating vortices at the rotor leading edge in close$_{2D}$ and far$_{2D}$. For far$_{2D}$, the clockwise vortex remains within the rotor core flow (away from the boundary layers), while the counter-clockwise vortex travels along the blade surface. For close$_{2D}$, both counter-rotating vortices remain within the core flow.

The circulation is:

\[ \Gamma = \int \omega dA, \]

where the vorticity is computed from velocity gradients using a centered-difference scheme. The normalized circulation is given in Table 4.1, which shows 2.8% difference between far$_{2D}$ and close$_{2D}$. The difference in circulation should be the same for the counter-clockwise vortices since there is a small net loading on the upstream vane row (the difference in the time-averaged flow angle at the vane trailing edge between the two configurations was measured to be 0.2 degrees). Measuring the circulation for the counter-clockwise rotating vortex at the rotor inlet is not possible for the far$_{2D}$ configuration because it is intersected by the rotor blade.

Table 4.1: Circulation for the clockwise vortex at the rotor trailing edge.

<table>
<thead>
<tr>
<th></th>
<th>( \Gamma/(\text{Tip Speed} \times c_x) )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Far$_{2D}$</td>
<td>0.1598</td>
</tr>
<tr>
<td>Close$_{2D}$</td>
<td>0.1517</td>
</tr>
<tr>
<td>% difference Far$<em>{2D}$ - Close$</em>{2D}$</td>
<td>2.8%</td>
</tr>
</tbody>
</table>
To assess the difference in vortex size, a vorticity-weighted radius is used, defined as:

\[
\tilde{r}^\omega = \frac{\int r \omega dA}{\int \omega dA},
\]

(4.5)

where the radial coordinate \( r \) is measured from the location of maximum vorticity. The value of \( \tilde{r}^\omega \) represents a vortex of constant vorticity and radius \( \tilde{r}^\omega \), outside which the vorticity is zero, as in Figure 4.4.

![Figure 4.4: Vortex equivalent radius, (a) distribution of vorticity as a function of radius, and (b) equivalent radius to describe the distribution of vorticity depicted in (a).](image)

Table 4.2 gives \( \tilde{r}^\omega \) for the clockwise vortices described by Table 4.1. The difference in radii for the two configurations is 10.4%.  

<table>
<thead>
<tr>
<th></th>
<th>( \tilde{r}^\omega / \text{pitch} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \text{Far}_2\text{D} )</td>
<td>0.0625</td>
</tr>
<tr>
<td>( \text{Close}_2\text{D} )</td>
<td>0.0566</td>
</tr>
<tr>
<td>% difference ( \text{Far}_2\text{D} - \text{Close}_2\text{D} )</td>
<td>10.4%</td>
</tr>
</tbody>
</table>

Table 4.2: Circulation-weighted radii for the clockwise vortex at the rotor trailing edge.

The small effect of differences in the circulation and size of the shed vortices on performance differences between \( \text{far}_2\text{D} \) and \( \text{close}_2\text{D} \) are confirmed by comparing the non-
uniformity in the velocity field at the rotor leading edge. A measure of the non-uniformity at any given plane is the quantity:

$$\frac{1}{2} \rho u'^2,$$  \hspace{1cm} (4.6)

where $u'^2$ is defined as:

$$u'^2 = (u_x - \bar{u}_x)^2 + (u_y - \bar{u}_y)^2.$$  \hspace{1cm} (4.7)

The overbar represents the time- and pitch-wise area- average of the velocity component. With no pressure variations within the flow, the quantity in (4.6) represents the potential for stagnation pressure loss in an incompressible flow, as derived in Appendix A. In the case of interest, there are pitch-wise non-uniformities in the velocity due to the rotor pressure field, but the pressure field is assumed to change the non-uniformity metric by the same magnitude in both configurations.

The non-uniformity metric, averaged over the pitch and in time, and non-dimensionalized by the dynamic head at the inlet is:

$$\frac{1}{2} \rho u'^2 \left[ \frac{1}{\tau A_{inlet}} \int \int \rho u'^2 \delta A dA dt \right] = \frac{P_{t,inlet} - P_{inlet}}{P_{t,inlet} - P_{inlet}}.$$

The quantity in (4.8) is plotted in Figure 4.5 as a function of axial location. In Figure 4.5a, the difference in the non-uniformity metric differs by 0.2% between the two configurations at the rotor leading edge. In Figure 4.5b, the curve for close2D is shifted to the right so that the vanes in both configurations are aligned at the same axial location, as opposed to the rotor blades being aligned in Figure 4.5a. It can be seen that the non-uniformity metric follows similar trends up to the rotor leading edge for both configurations. However, the trend within the rotor and downstream differs between the
two configurations, indicating different flow fields within the rotor. The differences within the rotor are due to the different vortex trajectories, since this is the only differing flow feature entering the rotor. At the rotor trailing edge, the difference in the non-uniformity metric between configurations is 23%, much greater than at the leading edge. In summary, the rotor is subjected to a similar non-uniformity at the leading edge in both configurations.

With a similar non-uniformity ahead of the rotor, the performance differences between far2D and close2D can be linked to how the vortices are processed within the rotor, due to their relative respective trajectories. This is discussed in the following section.
Figure 4.5: Non-uniformity metric of equation 4.8 as a function of axial position with (a) the rotor blades in both configurations at the same axial location, (b) the vanes at the same axial location.
4.4 Rotor Response to a Change in Vortex Trajectory

The rotor behavior due to a change in vortex trajectory is first assessed in terms of the difference in work input and efficiency. Although there is a lower work input in close\textsubscript{2D} compared to far\textsubscript{2D}, the efficiency at the rotor trailing edge is similar for the two cases. For close\textsubscript{2D}, the lower work input (lower total temperature rise), is accompanied by a lower entropy rise with respect to far upstream. However, the difference in entropy generation between the two configurations does not occur within the rotor. Instead, more entropy is generated in far\textsubscript{2D} from the inlet of the domain to the rotor leading edge plane due to higher shock losses from a higher operating back pressure. (Although there are shocks of different strengths in both configurations, the vortices are unaffected by any possible interactions with them, as evidenced by similar values of the non-uniformity metric at the rotor leading edge plane, as in Section 4.3).

The overall entropy rise within the rotor differs by 0.6% between far\textsubscript{2D} and close\textsubscript{2D}, as seen in time-averaged values in Table 4.3. The entropy increase within the rotor is calculated inside and outside the boundary layer, i.e. the core flow which includes the shock waves, using the computational dissipation described in Chapter 3. For control volumes encompassing these two regions, the reversible heat transfers across the surface are negligible compared to the irreversible viscous and thermal dissipation within the volume, and the integration of the computational dissipation is therefore equal to the entropy generated. The distribution of entropy generation within the rotor passage in both configurations is different, because there are two vortices in the core flow for close\textsubscript{2D}, and one vortex in the core flow in far\textsubscript{2D}. As seen in Table 4.3, there is a 0.4% difference in the entropy generated within the boundary layer control volume, and 0.9%
difference in the core flow. The shock losses (also given in the table) are found to be higher in \textit{far}_{2D} than \textit{close}_{2D} by 6.8\%. In summary, the overall entropy rise in the rotor for both configurations is similar because there are higher shock losses in \textit{far}_{2D} but less mixing losses (from one vortex in the core flow for \textit{far}_{2D}, versus two vortices in \textit{close}_{2D}).

Note that the difference in shock losses are of the same magnitude as the differences in losses from the vortices in the core flow, which is not the case for the three-dimensional configurations described in Chapter 3. The reason for this is that the difference in blade row spacing for the three-dimensional configurations is six times larger than the difference in the two-dimensional configurations. Thus, the difference in the strength of the vortices is also much greater, and a larger difference in entropy generation from the vortices compared to the difference in shock losses.

Table 4.3: Time-averaged entropy generation in the specified control volumes within rotor.

<table>
<thead>
<tr>
<th></th>
<th>Overall</th>
<th>Boundary layer</th>
<th>Core flow</th>
<th>Rotor shock</th>
</tr>
</thead>
<tbody>
<tr>
<td>\textit{Far}_{2D}</td>
<td>0.0925</td>
<td>0.0625</td>
<td>0.0300</td>
<td>0.0521</td>
</tr>
<tr>
<td>\textit{Close}_{2D}</td>
<td>0.0931</td>
<td>0.0628</td>
<td>0.0303</td>
<td>0.0488</td>
</tr>
<tr>
<td>\textit{Close}<em>{2D} - \textit{Far}</em>{2D} % difference</td>
<td>0.6%</td>
<td>0.4%</td>
<td>0.9%</td>
<td>-6.8%</td>
</tr>
</tbody>
</table>

While the entropy generation within the rotor is similar between the two configurations, the different vortex trajectories within the rotor cause the entropy generation downstream of the rotor to be different, leading to a 0.3 point difference in efficiency far downstream. The differences in the flow field at the rotor trailing edge plane are the number of vortices in the core flow and the sizes of the boundary layers. The effect of the different boundary layers on the difference in entropy generation downstream of the rotor for the two configurations is now assessed.
In Figure 4.6, the red color marks areas where the close\textsubscript{2D} boundary layer is larger than the far\textsubscript{2D} boundary layer, and the blue color marks where the far\textsubscript{2D} boundary layer is larger. The boundary layer is defined as the location with the computational dissipation (based on (3.3)) at least two orders of magnitude higher than the core flow.

Three boundary layer characteristic quantities are calculated at the rotor trailing edge, for both pressure and suction sides. These are the displacement thickness

\[ \delta^* = \int_0^{\gamma_E} \left( 1 - \frac{u_x}{u_E} \right) dy_E, \quad (4.9) \]

the momentum thickness

\[ \theta = \int_0^{\gamma_E} \left( 1 - \frac{u_x}{u_E} \right) \rho u_x \frac{u_x}{\rho E u_E} dy_E, \quad (4.10) \]

and the energy thickness

\[ \chi = \int_0^{\gamma_E} \left( 1 - \frac{u_x}{u_E} \right) \rho u_x \frac{u_x}{\rho E u_E} dy_E, \quad (4.11) \]
In these, the subscript $E$ represents the free stream condition. The displacement thickness and momentum thickness are larger in close$2D$ than far$2D$ by 13% and 3% respectively. The energy thickness is greater in far$2D$ than close$2D$ by 4%. A larger boundary layer in close$2D$ is consistent with a reduced work input compared to far$2D$, although a direct connection between the displacement thickness and work input has not made. The mechanism that leads to the different work inputs between the configurations is under investigation by Ph.D. candidate, Sean Nolan (M.I.T.).

The entropy generation from the mixing of the boundary layers from the rotor trailing edge plane to far downstream is assessed to determine its contribution to the total entropy generated in the same region in both configurations. The entropy rise from the boundary layers has been estimated assuming a uniform core flow, and solving the two-dimensional conservations equations using the quantities calculated from equations (4.9)-(4.11). The procedure is outlined in Appendix B. The overall entropy rise generated from the rotor exit to far downstream, and the entropy rise from the mixing of the boundary layers alone, are given in Table 4.4. The difference in the boundary layer entropy rise accounts for 40% of the difference in the entropy rise between close$2D$ and far$2D$. Therefore, the effect of vortex trajectory on the entropy generation downstream of the rotor is at least partly due to its effect on the rotor boundary layers. Moreover, there was less entropy generated downstream of the rotor when one vortex was located near the blade surface, which makes the vortex trajectory in far$2D$ favorable to that in close$2D$.
Table 4.4: Entropy rise from rotor trailing edge to far downstream.

<table>
<thead>
<tr>
<th></th>
<th>Overall</th>
<th>Boundary Layer</th>
</tr>
</thead>
<tbody>
<tr>
<td>Close$_{2D}$</td>
<td>0.0568</td>
<td>0.0233</td>
</tr>
<tr>
<td>Far$_{2D}$</td>
<td>0.0489</td>
<td>0.0203</td>
</tr>
<tr>
<td>Difference (Close$<em>{2D}$-Far$</em>{2D}$)</td>
<td>0.0078</td>
<td>0.0030</td>
</tr>
</tbody>
</table>

The remaining difference in the entropy rise measured far downstream must be due to the mixing and diffusion to a final uniform state of two vortices in the core flow for close$_{2D}$ versus one vortex for far$_{2D}$. The difficulty in measuring the entropy generation from the vortices directly from the 2D simulations is isolating them from the rest of the flow field. Downstream of the rotor trailing edge, the rotor boundary layer and vortices mix together so that isolating the vortex at each instant in time while following it downstream becomes difficult. Further, a defining characteristic associated with the vortices was not obvious, since these characteristics were also associated with other parts of the flow field (for example, using vorticity to mark the vortices would also capture regions of the rotor boundary layer). Marking the fluid particles associated with the vortices from their inception at the vane trailing edge is also difficult. Future work is thus suggested to conclusively demonstrate that the mixing of the vortices in the core flow accounts for the remaining difference in entropy generation.

4.5 Chapter 4 Summary

This chapter describes the impact of vortex trajectory within a rotor on its performance. Two configurations are studied which have an axial spacing between the vane and rotor that differ by 11%. The change in blade row spacing results in two different vortex trajectories within the rotor. In one, there are two vortices in the passage
core flow. In the other, one vortex is within the passage core flow, and the other travels along the surface of the blade.

It is found that the configuration with two vortices in the core flow has a lower work input and lower efficiency. The difference in the rotor performance is attributed to the difference in vortex trajectory through arguments that the flow non-uniformity ahead of the rotor is similar for both spacings. Roughly 40% of the difference in the downstream efficiency is attributed to a difference in the boundary layer at the rotor trailing edge.
Chapter 5

Summary and Conclusions

- The impact of axial blade row spacing on rotor performance for a highly-loaded, high Mach number single stage compressor has been assessed using time accurate, three-dimensional computations for two axial blade row spacings. At the same mass flow rate, the reduced spacing had a 0.7 point lower efficiency and a 1% lower work input than the larger spacing.

- A numerical technique was developed to accurately quantify entropy generation from computational simulations, even in regions with high spatial high gradients, such as shock waves. The technique used the divergence of velocity as a marker to precisely define the shock region.

- The dominant entropy generating mechanism that leads to the performance differences between the configurations is associated with the vortices which are shed from the upstream vanes due to the rotor pressure field.

- The vortex trajectory within the rotor, which is a function of blade row spacing, impacts rotor performance. Two-dimensional computations were carried out for two axial blade row spacings with different vortex trajectories. In one
configuration, one shed vortex was located within the boundary layer and one outside the boundary layer (in the core flow). In the other configuration, both vortices were located in the core flow. The latter configuration had a 3% lower work input and a 0.3 point difference in efficiency, as measured far downstream of the rotor.

- For a two-dimensional geometry at a given Mach number, the vortex trajectory is found to be a function of the ratio between the convective time scale for the vortices to travel the length of the axial gap between the vane and rotor blade rows, and the rotor period (i.e. the time for a rotor to move one rotor pitch).

- The largest difference in the entropy generation between the two configurations investigated occurred downstream of the rotor trailing edge. This is due to different mixing processes that result from differences in the rotor boundary layer and the different number of the vortices located in the core flow.
Chapter 6

Suggested Future Work

At the end of Chapter 4, future work was suggested to quantify the entropy generation associated with the shed vortices downstream of the rotor trailing edge. This is mentioned here again to reiterate that by using a different approach to calculate the differences in entropy generation from the number of vortices, the boundary layer calculations could also be verified and supported. The calculation of the entropy rise from the mixing of the boundary layers downstream of the rotor invokes the assumption that the flow field outside of the boundary layer is uniform. The impact of this assumption has not been assessed in the mixing calculations outlined in Appendix B.

Also to be assessed is the impact of change in blade row spacing on the vortex trajectory within the rotor, at each radial span, for a three-dimensional flow. A change in vortex trajectory at each radial plane may be detrimental or beneficial to the overall performance. The two-dimensional description of the effect of vortex trajectory on performance, as presented in this thesis, is applicable at each radial plane if three dimensional effects are unimportant. Recent work by Turner et al [20] suggests that radial migration of high entropy fluid from the tip region impacts the radial profile of loss downstream of the rotor. The relative importance of a change in vortex trajectory with
respect to other three-dimensional flow mechanisms that effect rotor performance as blade row spacing is varied has yet to be determined.

As suggested at the end of Section 1.2, a blade row spacing may exist that optimizes performance, due to competing effects of a change in vortex trajectory and change in vortex strength with changes in blade row spacing. An investigation into this idea is suggested for future work.
References


Appendix A

Non-uniformity Metric

In Chapter 4, a metric to measure the degree of non-uniformity ahead of the rotor was defined as:

\[
\frac{1}{2} \rho u'^2 ,
\]  \hspace{1cm} (4.6)

where \( u'^2 \) is a perturbation velocity, defined as:

\[
u'^2 = (u_x - \bar{u}_x)^2 + (u_y - \bar{u}_y)^2 .
\] \hspace{1cm} (4.7)

The overbar represents the time- and pitch-wise area-average of the velocity component. The non-uniformity metric is averaged over the pitch and in time, and non-dimensionalized by the dynamic head at the inlet. Its mathematical form is:

\[
\frac{1}{2} \frac{\rho u'^2}{P_{t,\text{inlet}} - P_{\text{inlet}}} = \frac{1}{\tau A_{\text{total}}} \left[ \int \frac{1}{2} \rho u'^2 dA \right] dt .
\] \hspace{1cm} (4.8)

The averaged quantity in (4.8) will now be shown to be a suitable metric to define the non-uniformity in the flow field. This is first shown by demonstrating that the non-uniformity metric represents the pressure difference between two axial planes by using the conservation of momentum (however, the factor of \( \frac{1}{2} \) does not appear). Using linear momentum, the non-uniformity in the flow field can be shown to act as a blockage that decreases the flow area, and hence, reduces the pressure. The non-uniformity can also be
related to the stagnation pressure losses if the flow at an axial location was allowed to mix to a uniform state.

First, linear momentum is used to determine how non-uniformity affects pressure changes in a flow. The impulse function is written as:

\[ I = p + \rho u^2. \]  

(A.1)

The impulse function, when integrated across the inlet and exit flow area (planes 'a' and 'b') of a specified control volume, is the same if there are no forces acting on the control volume and there is no change in area. Steady flow is assumed. Mathematically, this is stated as:

\[ \Delta I = I_b - I_a = 0 \]  

(A.2)

The overbar in this equation, and in the rest of this section, represents the pitch-wise or area-average of the impulse function. Pressure and velocity at any point can be defined as the sum of its average quantity on the plane of interest, plus its spatially-varying quantity:

\[ p = \overline{p} + p' \]  

(A.3)

\[ u = \overline{u} + u' \]  

(A.4)

Note that the area-average of the spatially-varying component of the pressure and velocity are zero, i.e. \( \overline{p'} = 0 \) and \( \overline{u'} = 0 \).

Substituting the expressions of A.3 and A.4 into the impulse function:

\[ I = \overline{p} + p' + \rho(\overline{u}^2 + 2\overline{u}u' + u'^2) \]  

(A.4)
By assuming the density and cross-sectional area remains constant, the continuity equation simplifies to: $\overline{u_a} = \overline{u_b}$. The difference in the momentum between planes ‘a’ and ‘b’ is:

$$\Delta I = \overline{\rho b - \rho a} + \rho' b - \rho' a + 2 \overline{\rho} (u_b' - u_a') + \rho(u_b'^2 - u_a'^2)$$  \hspace{1cm} (A.5)

Area-averaging the impulse function gives:

$$\overline{\Delta I} = \overline{\rho b - \rho a} + \rho\overline{(u_b'^2 - u_a'^2)} = 0$$  \hspace{1cm} (A.6)

The difference in pressure from planes ‘a’ to ‘b’ is therefore related to the non-uniformity at both stations by rearranging (A.6):

$$\overline{\rho b - \rho a} = -\rho\overline{(u_b'^2 - u_a'^2)} = -\rho \Delta u'^2$$  \hspace{1cm} (A.7)

If at plane ‘b’, the flow is fully mixed out, then:

$$\overline{\rho b - \rho a} = \rho \overline{u_b'^2}$$  \hspace{1cm} (A.8)

The term on the right-hand side of A.8 is similar to the non-uniformity metric, and therefore indicates that the metric represents the pressure rise that would be obtained if the non-uniformity was allowed to mix to a uniform state. Therefore, the non-uniformity represents a blockage in the flow area, which tends to reduce pressure.

The $\frac{1}{2}$ term included in the non-uniformity metric appears when considering the difference in mass-averaged stagnation pressure differences between planes ‘a’ and ‘b’.

The stagnation pressure for incompressible flow is given as:

$$p_i = p + \frac{1}{2} \rho u^2$$  \hspace{1cm} (A.9)

The stagnation pressure can be multiplied by the velocity and averaged across the pitch to find a velocity-weighted average value for the stagnation pressure. This is equivalent to multiplying the stagnation pressure by the local mass flux and finding a mass-averaged
stagnation pressure. However, the density is not included when multiplying by the mass
flux because the density is assumed constant, and therefore is not required when
comparing two axial locations. The velocity-weighted stagnation pressure is written as:

\[ u_{p_1} = u_p + \frac{1}{2} p u^3 \quad (A.10) \]

Substituting the expressions for pressure and velocity from (A.3) and (A.4) into (A.10):

\[ u_{p_1} = \bar{u} \bar{p} + \bar{u} p' + u' \bar{p} + u' p' + \frac{1}{2} \rho \left( \bar{u}^3 + 3 \bar{u}^2 u' + 3 \bar{u} u'^2 + u'^3 \right) \quad (A.11) \]

A pitch-wise average of (A.11) gives:

\[ \bar{u}_{p_1} = \bar{u} p + \bar{u} p' + \frac{1}{2} \rho \bar{u}^3 + \frac{3}{2} \rho \bar{u} \bar{u} u'^2 + \frac{1}{2} \rho \bar{u} u'^3 \quad (A.12) \]

The difference in mass-averaged stagnation pressure between planes ‘a’ and ‘b’ becomes
(by also applying the continuity equation):

\[ \Delta u_{p_1} = \Delta \bar{u} \bar{p} + \Delta \bar{u} \bar{p}' + \frac{3}{2} \rho \bar{u} \Delta \bar{u} u'^2 + \frac{1}{2} \rho \Delta \bar{u} u'^3 \quad (A.13) \]

Using the pressure difference from conservation of linear momentum given in (A.7), the
first term on the left-hand side of (A.13) can be replaced:

\[ \Delta u_{p_1} = \bar{u} \left( -\rho \Delta \bar{u} u'^2 \right) + \Delta \bar{u} \bar{p}' + \frac{3}{2} \rho \bar{u} \Delta \bar{u} u'^2 + \frac{1}{2} \rho \Delta \bar{u} u'^3 \quad (A.14) \]

which reduces to:

\[ \Delta u_{p_1} = \Delta \bar{u} \bar{p}' + \frac{1}{2} \rho \bar{u} \Delta \bar{u} u'^2 + \frac{1}{2} \rho \Delta \bar{u} u'^3 \quad (A.15) \]

For a non-uniform flow that does not have large variations in the velocity profile, the
approximation \( \bar{u} \Delta \bar{u} u'^2 \gg \Delta \bar{u} u'^3 \) is valid. Furthermore, for one-dimensional shear flow,
\( \rho' = 0 \). The expression in (A.15) becomes:
\[ \Delta \overline{u} \overline{p} = \overline{u} \left( \frac{1}{2} \rho \overline{u}^2 \right) \]  
(A.16)

Therefore, the non-uniformity metric at any plane represents the potential for stagnation pressure losses if the flow at that axial location was allowed to mix to a uniform state in a constant area duct. The assumptions include incompressible, one-dimensional flow, which makes the use of the non-uniformity metric approximate.

Finally, the non-uniformity metric also represents the difference between the area-averaged stagnation pressure and the stagnation pressure based on the average quantities on an axial plane. The latter stagnation pressure is given as:

\[ \hat{\overline{p}}_t = \overline{p} + \frac{1}{2} \rho \overline{u}^2 \]  
(A.17)

An area-average of the stagnation pressure \( \overline{p}_t^A \) is (since area-average of \( u' \) and \( \rho' \) are zero):

\[ \overline{p}_t^A = \overline{p} + \frac{1}{2} \rho \left( \overline{u}^2 + \overline{u'^2} \right) \]  
(A.18)

And the difference between (A.17) and (A.18) also leads to the expression of the non-uniformity metric:

\[ \overline{p}_t^A - \hat{\overline{p}}_t = \frac{1}{2} \rho \overline{u}^2 \]  
(A.19)

In summary, the non-uniformity metric calculated in (4.8) at any axial plane of interest represents a number of physical quantities: the blockage at that location, the potential for stagnation pressure losses, and the difference in the area-averaged stagnation pressure compared to the stagnation pressure compared from the average values of pressure and velocity at that location. All three physical scenarios are an indication of the level of non-uniformity at a specified location.
The analysis of the present section assumed that the flow non-uniformity was constant in time. However, since the flow field at any axial location is unsteady, the non-uniformity metric is adjusted by computing the perturbation velocity given in (4.7) with respect to the time- and pitch-wise average of the velocity component. This makes the non-uniformity metric an approximation, but nonetheless provides a suitable quantity when comparing similar flow fields for the two configurations under study in Chapter 4.
Appendix B

Mixing of Boundary Layers to a Uniform Flow State

In Chapter 4, two-dimensional calculations were conducted to study the effect of a change in vortex trajectory on rotor performance. Two configurations under examination were referred to as “far2D” and “close2D.” At the rotor trailing edge plane, the boundary layer characteristic quantities are measured from (4.9)-(4.11) for both configurations. The present discussion outlines the procedure to determine the contribution of the boundary layers to the entropy rise from the rotor trailing edge plane to far downstream, where the flow field is uniform.

Figure B.1: Schematic that represents the mixing of a swirling flow to a uniform flow state.
To determine the entropy rise from the boundary layers alone, the first assumption is that the core flow is uniform in the pitch-wise direction, as seen in Figure B.1. Then the conservation equations are applied from the rotor trailing edge plane to far downstream, allowing the boundary layers to mix with the core flow to a uniform flow state. The conservation equations include conservation of mass, linear and angular momentum, and energy. A swirl angle is also measured at the rotor exit and included in the mixing calculations to better represent the flow field. Other required quantities to define the rotor exit state of are the time-averaged mass flow rate, as well as the mass- and time-averaged pressure, and entropy flux. The temperature and density are found from the constitutive relation for entropy.

A control volume is placed from the rotor exit to far downstream. The conservation equations are:

\[ \dot{m} = \rho u_t (A - \delta^*) \cos \alpha_i = \rho u_{\infty} A \cos \alpha_{\infty} \]  
\[ p_i A + \rho u_i^2 \cos^2 \alpha_i (A - \theta - \delta') = p_{\infty} A + \rho u_{\infty}^2 \cos^2 \alpha_{\infty} A \]  
\[ \rho u_i^2 \sin \alpha_i \cos \alpha_i (A - \theta - \delta') = \rho u_{\infty}^2 \sin \alpha_{\infty} \cos \alpha_{\infty} A \]  
\[ c_p T_i \dot{m} + \frac{\rho u_i^4}{2} \cos \alpha_i (A - \delta^* - \theta') = \left( c_p T_{\infty} + \frac{u_{\infty}^2}{2} \right) \dot{m} \]

The ideal gas law is also used:

\[ p = \rho R T \]

The constitutive relation to calculate the entropy rise is:

\[ \frac{\Delta s}{R} = \frac{\gamma}{\gamma - 1} \ln \left( \frac{T_{\infty}}{T_h} \right) - \ln \left( \frac{p_{\infty}}{p_h} \right) \]
From mass conservation in (B.1), the uniform core velocity, $u_t$, can be found since the mass flow rate is measured, and all other quantities are known. The values at downstream infinity can then be determined by solving (B.1)-(B.5) simultaneously, and the entropy rise calculated from (B.6).

The entropy increase from the rotor leading edge to downstream infinity was independent of the value of the entropy flux at the rotor leading edge. There was a slight dependence of the entropy rise due to mixing on pressure. Varying the pressure at the rotor exit by 1.3% (which is the difference between the mass-averaged and area-averaged pressure) caused the difference in entropy rise between the two configurations to change by 2.5%. This difference in the entropy rise was not enough to change the conclusions derived from the mixing calculation. The results of the mixing calculations are presented in Table 4.4.