Numerical Analysis on the Effects of Blade Loading on Vortex Shedding and Boundary Layer Behavior in a Transonic Axial Compressor

Kenneth Phillip Clark
Brigham Young University - Provo

Follow this and additional works at: https://scholarsarchive.byu.edu/etd

Part of the Mechanical Engineering Commons

BYU ScholarsArchive Citation

This Thesis is brought to you for free and open access by BYU ScholarsArchive. It has been accepted for inclusion in Theses and Dissertations by an authorized administrator of BYU ScholarsArchive. For more information, please contact scholarsarchive@byu.edu, ellen_amatangelo@byu.edu.
Numerical Analysis on the Effects of Blade Loading on Vortex Shedding and Boundary Layer Behavior in a Transonic Axial Compressor

Kenneth P. Clark

A thesis submitted to the faculty of Brigham Young University in partial fulfillment of the requirements for the degree of Master of Science

Steven E. Gorrell, Chair
Daniel Maynes
Scott L. Thomson

Department of Mechanical Engineering Brigham Young University
August 2011

Copyright © 2011 Kenneth P. Clark All Rights Reserved
ABSTRACT

Numerical Analysis on the Effects of Blade Loading on Vortex Shedding and Boundary Layer Behavior in a Transonic Axial Compressor

Kenneth P. Clark
Department of Mechanical Engineering, BYU
Master of Science

Multiple high-fidelity, time-accurate computational fluid dynamics simulations were performed to investigate the effects of upstream stator loading and rotor shock strength on vortex shedding characteristics in a single stage transonic compressor. Various configurations of a transonic axial compressor stage, including three stator/rotor axial spacings of close, mid, and far in conjunction with three stator loadings of decreased, nominal, and increased were simulated in order to understand the flow physics of transonic blade-row interactions. Low-speed compressors typically have reduced stator/rotor axial spacing in order to decrease engine weight, and also because there is an increase in efficiency with reduced axial spacing. The presence of a rotor bow shock in high-speed compressors causes additional losses as the shock interacts with the upstream stator trailing edge. This research analyzes the strength of shock-induced vortices due to these unsteady blade-row interactions.

The time-accurate URANS code, TURBO, was used to generate periodic, quarter annulus simulations of the Blade Row Interaction compressor rig. Both time-averaged and time-accurate results compare well with experimentally-observed trends.

It was observed that vortex shedding was synchronized to the passing of a rotor bow shock. “Normal” and “large” shock-induced vortices formed on the stator trailing edge immediately after the shock passing, but the “large” vortices were strengthened at the trailing edge due to a low-velocity region on the suction surface. This low velocity region was generated upstream of mid-chord on the suction surface from a shock-induced thickening of the boundary layer or separation bubble, due to the rotor bow shock reflecting off the stator trailing edge and propagating upstream. The circulation of the shock-induced vortices increased with shock strength (decreased axial spacing) and stator loading.

Most design tools do not directly account for unsteady effects such as blade-row interactions, so a model is developed to help designers account for shock-induced vortex strength with varying shock strength and stator loading. An understanding of the unsteady interactions associated with blade loading and rotor shock strength in transonic stages will help compressor designers account for unsteady flow physics early in the design process.

Keywords: Kenneth Clark, turbomachinery, gas turbine engines, transonic compressor, blade row interactions, vortex shedding, stator loading, suction side boundary layer, circulation
ACKNOWLEDGMENTS

The author would like to thank his advisor, Dr. Steve Gorrell, for all his help with this research and patience in both the research and writing processes. The author would also like to acknowledge the support of the Fulton Supercomputing Lab (FSL) and the Air Force Research Laboratory (AFRL) DoD Supercomputing Research Center (DSRC) for allowing these numerical simulations to be performed on their high performance supercomputing clusters, as well as their support in code setup, debugging, and hardware fixes. The author would also like to thank Dave Car for his help with post-processing code development, and Mike List for help with the pre-processing and the gridding. The author would also like to thank his wife, Kari Clark, for her endless support and patience.
# TABLE OF CONTENTS

### LIST OF TABLES


### LIST OF FIGURES


### NOMENCLATURE


### Chapter 1  Introduction

1.1 Motivation ......................................................... 3
1.2 Plan of Development ........................................... 5

### Chapter 2  Literature Review and Background

2.1 Wake Recovery ..................................................... 7
2.2 Blade-Row Interactions ........................................ 9
2.3 Vortex Shedding .................................................. 14
2.4 Stage Matching Investigation (SMI Rig) .......................... 17
2.5 Blade-Row Interaction (BRI) Rig ................................. 21
    2.5.1 BRI Geometry & Hardware ................................. 22
    2.5.2 BRI Analyses ............................................... 24
2.6 Summary ....................................................... 26

### Chapter 3  Numerical Setup

3.1 Computational Domain ........................................... 29
3.2 Grid Generation .................................................. 30
3.3 Initialization and Grid Partitioning .............................. 33
3.4 Numerical Solutions ............................................ 36
3.5 Simulation Convergence ........................................ 37

### Chapter 4  Results

4.1 Time-Averaged Efficiency ...................................... 43
4.2 Stator Loading ................................................... 45
    4.2.1 Time-Averaged ............................................. 46
    4.2.2 Unsteady .................................................... 47
4.3 Suction Surface Boundary Layer Behavior ....................... 49
    4.3.1 Boundary Layer at Nominal Loading ....................... 51
    4.3.2 Boundary Layer at Decreased Loading .................... 53
    4.3.3 Effects of Pressure Gradient ............................ 53
4.4 Vortex Shedding ................................................ 59
    4.4.1 “Normal” Vortex Formation ................................ 59
    4.4.2 “Large” Vortex Formation ................................ 63

### Chapter 5  Circulation Analysis and Model Development

5.1 Circulation ..................................................... 69
5.2 Dimensional Analysis ........................................... 74
5.3 Model Development ................................................................. 80

Chapter 6 Discussion of Results ..................................................... 85
  6.1 Recommendations for Future Work ........................................... 88

Chapter 7 Conclusion .................................................................... 91

REFERENCES ................................................................................. 95

Appendix A Vorticity Images .......................................................... 101

Appendix B Grid Partitioning .......................................................... 109

Appendix C Mass Flow Rate Histories .............................................. 111
  C.1 Mid, Decreased Loading ...................................................... 111
  C.2 Mid, Nominal Loading ......................................................... 111
  C.3 Mid, Increased Loading ....................................................... 112
  C.4 Far, Decreased Loading ....................................................... 113
  C.5 Far, Nominal Loading ......................................................... 113
  C.6 Close, Decreased Loading .................................................... 114

Appendix D TURBO Input Files ...................................................... 123
  D.1 Input Files ............................................................................ 123
    D.1.1 input00 ......................................................................... 123
    D.1.2 input01 ......................................................................... 126
    D.1.3 input02 ......................................................................... 126
    D.1.4 input03 ......................................................................... 126
LIST OF TABLES

2.1 BRI Aerodynamic Design Parameters (Note that the Stator is not Modeled in the Current Simulations) ................................................................. 24
2.2 Radius of Shock-Induced Vortices Located 20% Chord Length Downstream of Deswirler TE [16]. .................................................. 25
2.3 Circulation of Vortices Located 20% Chord Downstream of Deswirler TE [16]. ........ 26
3.1 Node Counts for Each Blade Passage in the Current Numerical Simulations. .......... 34
3.2 Normalized Values of Initialized Flow Field. \( p_{\text{ref}}=101.325 \text{ kPa}, T_{\text{ref}}=288.15 \text{ K} \) .......... 34
3.3 Partitioning Information for Mid Spacing at 0.0° Stagger (Nominal Loading) Resulting in 522 Partitions. ................................................. 35
3.4 Mass Flow Rates at Peak Experimental Efficiency in kg/s. ............................ 38
4.1 Inlet Average Static Pressure, \( p_1 \), Values in kPa. ........................................ 47
4.2 Inlet Average Total Pressure, \( p_{\text{t1}} \), Values in kPa. ............................... 47
5.1 Average “Normal” Vortex Strength for Shock-Induced Vortices with Units of \( m^2/s \). 70
5.2 Average “Large” Vortex Strength for Shock-Induced Vortices with Units of \( m^2/s \). 71
5.3 Average Circulation of All Shock-Induced Vortices (“Normal” and “Large”) in Units of \( m^2/s \). .......................................................... 72
5.4 Circulation for PIV Results of Shock-induced Vortices [16] in Units of \( m^2/s \). ...... 73
5.5 Percent Difference Between Experimental PIV Circulation Results (Table 5.4) and the Average of all the CFD Circulation Results (Table 5.3). .......... 73
5.6 Time-Averaged Diffusion Factor for Each Configuration. .............................. 75
5.7 Time-Averaged Shock Strength, \( \Delta p/p \), for Each Configuration. ................ 76
5.8 Time-Averaged Wake Displacement Thickness, \( \delta^* \), for Each Configuration (in mm). 79
5.9 Time-Averaged Suction Surface Boundary Layer Velocity for Each Configuration (in m/s). .................................................. 79
B.1 Partitioning Information for Mid Spacing at -3.0° Stagger (Decreased Loading) Resulting in 348 Partitions. ............................................ 109
B.2 Partitioning Information for Mid Spacing at 0.0° Stagger (Nominal Loading) Resulting in 522 Partitions. Note that Mid Spacing at +1.5° Stagger (Increased Loading) Results in the Same Partitions. .................. 109
B.3 Partitioning Information for Far Spacing at -3.0° Stagger (Decreased Loading) Resulting in 516 Partitions. ............................................ 110
B.4 Partitioning Information for Far Spacing at 0.0° Stagger (Nominal Loading) Resulting in 516 Partitions. ............................................ 110
B.5 Partitioning Information for Close Spacing at -3.0° Stagger (Decreased Loading) Resulting in 996 Partitions. ............................................ 110
B.6 Partitioning Information for Close Spacing at 0.0° Stagger (Nominal Loading) Resulting in 996 Partitions. Note that Close Spacing at +1.5° Stagger (Increased Loading) Results in the Same Partitions. .................. 110
## LIST OF FIGURES

2.1 Diagram Showing the Phenomenon of Wake Recovery, Causing $\Delta o < \Delta i$ Because of the Flow Acceleration. ................................................................. 8

2.2 Disturbance Velocities Showing “Negative Jet” Effect of a Passing Rotor Wake on the Downstream Stator SS BL [24]. ................................................................. 11

2.3 Shadowgraph Image Sequence Showing Shock-Induced Vortex Shedding in a Linear Cascade Test [14]. ................................................................. 17

2.4 Comparison of CFD and Shadowgraph Images of Shock-Induced Vortex Shedding in a Linear Cascade Test [37]. Note: Flow is from Right to Left. .............. 18

2.5 Comparison of CFD and PIV Absolute Velocity Contours Showing the SS Separation at the Stator TE [37]. Note: Flow is from Upper Right to Lower Left. .... 18

2.6 SMI Showing Entropy Flux at Close and Far Axial Spacing [12]. ....................... 19

2.7 Performance Speed Lines for the SMI, Showing the Worst Performance at Close, and the Best at Far [11]. ................................................................. 20

2.8 Cross-Section of Blade-Row Interaction (BRI) Rig. ........................................ 22

2.9 Midspan Slices of Blade-Row Interaction (BRI) Rig, Showing the Replacement of the Blunt Wake Generator with Swirler and Deswirler Rows. ...................... 22

2.10 Midspan Slices of the Swirler and Deswirler Rows for Decreased and Nominal loading ......................................................................................... 23

2.11 PIV Results [16] Showing Deswirler-Shed Vortices at Mid Spacing. ............ 27

3.1 The Computational Domain of the Simulations. ............................................. 30

3.2 Grid at Midspan Showing Two Blade Passages of Each Blade Row (Every 4th Node Shown). Coordinate Axis Values are in Meters. ................................. 31

3.3 Grid Details at Midspan Between the Deswirler and Rotor. Coordinate Axis Values are in Meters. ................................................................. 32

3.4 Grid Details of Deswirler TE at Midspan. Coordinate Axis Values are in Meters. . 33

3.5 Mass Flow Rate History for Far Spacing at Decreased Loading. Note: IT=Initial Transients, IC=Initial Convergence, LAC=Low-Accuracy Convergence, IA=Increased-Accuracy, DTS=Decreased Time Step. .................................. 39

3.6 Mass Flow Rate History for the Time-Averaged Solution Generation for Mid Spacing at Nominal Loading. ........................................................................... 41

3.7 Mass Flow Rate History for the Unsteady Solution Generation for Mid Spacing at Nominal Loading. ............................................................................. 42

4.1 Experimental and Time-Averaged Efficiency Speed Lines for Current Simulations of Mid Spacing at Decreased Loading (DL), Nominal Loading (NL), and Increased Loading (IL). CFD Results are Scaled by 0.95. .......... 45

4.2 Experimental and Time-Averaged Pressure Ratio ........................................... 46

4.3 Time-Averaged Pressure Coefficient on Deswirler Versus Chord Fraction at Midspan for Mid Spacing at Decreased Loading (DL), Nominal Loading (NL), and Increased Loading (IL). ......................................................... 48

4.4 Unsteady and Time-Averaged Pressure Coefficient on Deswirler Versus Chord Fraction at Midspan for Mid Spacing at Nominal Loading. .................. 49
4.5 Instantaneous Pressure Coefficient Over the Stator Chord for Mid Spacing at Nominal Loading. ................................................................. 50
4.6 Contours of Axial Velocity with Lines of Vorticity at Midspan for Nominal Loading, Showing the Formation of the Low-Velocity Region. .................................................. 52
4.7 Contours of Axial Velocity with Lines of Vorticity at Midspan for Decreased Loading. 54
4.8 Contours of Normalized Pressure Gradient Magnitude with Lines of Vorticity at Midspan for Nominal Loading. .................................................. 55
4.9 Contours of Normalized Pressure Gradient Magnitude (Same Levels as in Figure 4.8) with Lines of Vorticity at Midspan for Nominal Loading for the First 30% Deswirler Chord Showing the Effects of a Passing Pressure Wave on the Deswirler Pressure Side. ................................................................. 57
4.10 Contours of Normalized Pressure Gradient Magnitude (Same Levels as in Figure 4.8) with Lines of Vorticity at Midspan for Nominal Loading for the First 30% Deswirler Chord Showing the Effects of a Passing Pressure Wave on the Deswirler Suction Side. ................................................................. 57
4.11 Contours of Normalized Pressure Gradient Magnitude (Same Levels as in Figure 4.8) with Lines of Vorticity at Midspan for Nominal Loading for the First 30% deswirler chord Showing the Combined Effects of a Passing Pressure Wave on the Deswirler Pressure Side and Another on the Suction Side. ................................................................. 58
4.12 Axial Velocity Contours with Vorticity Lines Showing “Normal” Vortex Formation on Deswirler TE at Midspan for Mid Spacing at Decreased Loading. ................................................................. 60
4.13 Axial Velocity Contours with Vorticity Lines Showing “Normal” Vortex Formation on Deswirler TE at Midspan for Mid Spacing at Nominal Loading. ................................................................. 61
4.14 Axial Velocity Contours with Vorticity Lines Showing “Normal” Vortex Formation on Deswirler TE at Midspan for Mid Spacing at Increased Loading. ................................................................. 62
4.15 Axial Velocity Contours with Vorticity Lines Showing “Large” Vortex Formation on Deswirler TE at Midspan at Decreased Loading. ................................................................. 65
4.16 Axial Velocity Contours with Vorticity Lines Showing “Large” Vortex Formation on Deswirler TE at Midspan at Nominal Loading. ................................................................. 66
4.17 Axial Velocity Contours with Vorticity Lines Showing “Large” Vortex Formation on Deswirler TE at Midspan at Increased Loading. ................................................................. 67

5.1 Vorticity Lines and Contours with a Non-Circular Integration Path for the Circulation of a Large Vortex for Mid Spacing at Nominal Loading. This is the Same Vortex Seen in Figure 4.13e. ................................................................. 70
5.2 Measured Vortex Circulation for “Normal” and “Large” Shock-Induced Vortices for Mid Spacing at all Three Loadings. See Tables 5.1 and 5.2. ................................................................. 71
5.3 Measured Blade Loading (Force on Blade in N/m) and Diffusion Factor at Midspan. ................................................................. 76
5.4 A Sample Shock Extraction Line, at 20% Pitch Above the TE ................................................................. 77
5.5 A Sample Static Pressure Profile Across the Rotor Bow Shock Near the Deswirler TE at 20% Pitch Above the TE ................................................................. 78
5.6 Vortex Strength for the Average of All Shock-Induced Vortices. ................................................................. 80
5.7 Predicted Vortex Strength for “Large” Vortices. See Table 5.2. ................................................................. 81
5.8 Predicted Vortex Strength for All Shock-Induced Vortices. See Table 5.3. ................................................................. 82
A.1 Radial Vorticity Contours with Vorticity Lines Showing “Normal” Vortex Formation on Deswirler TE at Midspan for Mid Spacing at Decreased Loading. Compare to Figure 4.12. ......................................................... 102
A.2 Radial Vorticity Contours with Vorticity Lines Showing “Normal” Vortex Formation on Deswirler TE at Midspan for Mid Spacing at Nominal Loading. Compare to Figure 4.13. ......................................................... 103
A.3 Radial Vorticity Contours with Vorticity Lines Showing “Normal” Vortex Formation on Deswirler TE at Midspan for Mid Spacing at Increased Loading. Compare to Figure 4.14. ......................................................... 104
A.4 Radial Vorticity Contours with Vorticity Lines Showing “Large” Vortex Formation on Deswirler TE at Midspan for Mid Spacing at Decreased Loading. Compare to Figure 4.15. ......................................................... 105
A.5 Radial Vorticity Contours with Vorticity Lines Showing “Large” Vortex Formation on Deswirler TE at Midspan for Mid Spacing at Nominal Loading. Compare to Figure 4.16. ......................................................... 106
A.6 Radial Vorticity Contours with Vorticity Lines Showing “Large” Vortex Formation on Deswirler TE at Midspan for Mid Spacing at Increased Loading. Compare to Figure 4.17. ......................................................... 107
C.1 Mass Flow Rate History for Mid Spacing at Decreased Loading. ...................... 112
C.2 Mass Flow Rate History for the Time-Averaged Solution Generation for Mid Spacing at Decreased Loading. .......................................................... 113
C.3 Mass Flow Rate History for the Unsteady Solution Generation for Mid Spacing at Decreased Loading. .......................................................... 114
C.4 Mass Flow Rate History for Mid Spacing at Nominal Loading. ...................... 115
C.5 Mass Flow Rate History for the Time-Averaged Solution Generation for Mid Spacing at Nominal Loading. .......................................................... 115
C.6 Mass Flow Rate History for the Unsteady Solution Generation for Mid Spacing at Nominal Loading. .......................................................... 116
C.7 Mass Flow Rate History for Mid Spacing at Increased Loading. ...................... 116
C.8 Mass Flow Rate History for the Time-Averaged Solution Generation for Mid Spacing at Increased Loading. .......................................................... 117
C.9 Mass Flow Rate History for the Unsteady Solution Generation for Mid Spacing at Increased Loading. .......................................................... 117
C.10 Mass Flow Rate History for Far Spacing at Decreased Loading. ...................... 118
C.11 Mass Flow Rate History for the Time-Averaged Solution Generation for Far Spacing at Decreased Loading. .......................................................... 118
C.12 Mass Flow Rate History for the Unsteady Solution Generation for Far Spacing at Decreased Loading. .......................................................... 119
C.13 Mass Flow Rate History for Far Spacing at Nominal Loading. ...................... 119
C.14 Mass Flow Rate History for the Time-Averaged Solution Generation for Far Spacing at Nominal Loading. .......................................................... 120
C.15 Mass Flow Rate History for the Unsteady Solution Generation for Far Spacing at Nominal Loading. .......................................................... 120
C.16 Mass Flow Rate History for Close Spacing at Decreased Loading. ...................... 121
C.17 Mass Flow Rate History for the Time-Averaged solution Generation for Close Spacing at Decreased Loading. 121
C.18 Mass Flow Rate History for the Unsteady Solution Generation for Close Spacing at Decreased Loading. 122
NOMENCLATURE

**Thermodynamic Properties**

- \( p \): Pressure
- \( T \): Temperature
- \( R \): Gas constant for air
- \( C_p \): Coefficient of pressure
- \( \gamma \): Ratio of specific heats
- \( M \): Mach number

**Compressor Parameters**

- \( \alpha \): Flow incidence angle
- \( C_x \): Axial chord length
- \( r \): Radius
- \( \theta \): Tangential Angle
- \( S \): Pitch \((r\theta)\)
- \( t \): Trailing edge blade thickness
- \( \dot{m} \): Mass flow rate
- \( \omega \): Vorticity
- \( \Gamma \): Circulation
- \( t/T \): Time per rotor passing period
- \( \eta \): Efficiency
- \( U \): Axial velocity
- \( u \): Velocity in x-direction (also \( U \))
- \( v \): Velocity in y-direction
- \( w \): Velocity in z-direction
- \( V_\theta \): Tangential velocity
- \( V \): Velocity magnitude

**Abbreviations**

- RMS: Root-mean-square
- NPG: Normalized pressure gradient
- AFRL: Air Force Research Laboratory
- FSL: Fulton Supercomputing Laboratory
- SMI: Stage Matching Investigation test rig
- BRI: Blade Row Interaction test rig
- LE: Airfoil leading edge
- TE: Airfoil trailing edge
- SS: Airfoil suction surface
- PS: Airfoil pressure surface
- DL: Decreased stator loading (corresponding to -3.0° stagger)
- NL: Nominal stator loading (corresponding to 0.0° stagger)
- IL: Increased stator loading (corresponding to +1.5° stagger)
- TGS: Turbomachinery Gridding System
- mid_-30: Mid spacing at decreased loading (-3.0° stagger)
- mid_00: Mid spacing at nominal loading (0.0° stagger)
- mid_+15: Mid spacing at increased loading (+1.5° stagger)
clo_-30  Close spacing at decreased loading (-3.0° stagger)
clo_00   Close spacing at nominal loading (0.0° stagger)
clo_+15  Close spacing at increased loading (+1.5° stagger)
far_-30  Far spacing at decreased loading (-3.0° stagger)
far_00   Far spacing at nominal loading (0.0° stagger)

**Subscripts, superscripts, and other indicators**

- $[\ ]_t$ indicates total or stagnation property
- $[\ ]_s$ indicates static thermodynamic property (or no subscript)
- $[\ ]_{ref}$ indicates reference value of property
- $[\ ]_i$ indicates a property at the inlet
- $[\ ]_e$ indicates a property at the exit
- $[\ ]_1$ indicates a property at the deswirler inlet
- $[\ ]_2$ indicates a property at the deswirler exit
- $[\ ]_\infty$ indicates a freestream property
- $\sigma$ Standard deviation
CHAPTER 1. INTRODUCTION

High performance turbomachines typically are designed with highly loaded blade rows and decreased axial spacing, thereby exhibiting significant unsteady losses between blade rows not observed in low-speed turbomachines. These blade-row interactions, such as the interaction of a shock with a blade surface or a blade wake, are a significant source of unsteadiness in high-speed turbomachines. Most contemporary compressor design tools do not directly account for these significant unsteady effects, but rather rely on steady state, empirical, or semi-empirical tools. A better understanding of such phenomena is needed to identify the impact of unsteady aerodynamics on compressor performance, to develop and validate tools for measuring and modeling unsteady flows and to develop design tools that more accurately account for unsteady aerodynamics.

Three dimensional experiments and computational simulations are necessary to accurately predict compressor performance, especially in the transonic regime where the rotor leading edge shock accounts for the majority of the pressure rise and loss. Adamczyk [1] described the need for experimental and numerical work which focused on unsteady fluid mechanics and the impact on axial turbomachinery performance. He said that experimental results increase understanding of these unsteady flows and can also be used to verify results obtained from design tools. Adamczyk described a need for multi-stage design tools that do not rely on empirical formulations or data as inputs. He showed that in order to develop design tools that account for unsteady characteristics, a more complete understanding of unsteady flows that are classified as nondeterministic, but are not turbulent in nature, must be obtained. An example of these unsteady flow characteristics is the shedding of vortices from a blade's trailing edge.

Vortex shedding in turbomachines has been the focus of research for some time. Hathaway [2] measured vortex shedding in fan rotors, which were shown to lead to spanwise redistribution of entropy by Kotidis and Epstein [3]. It has also been observed that the stretching
of vortices leads to flow instabilities which resulted in rapid mixing [4]. These unsteady non-turbulent flows appear to lead to the mixing of shear layers and therefore generate loss which reduces pressure rise and efficiency.

The impact of such mixing processes on aerodynamic performance of multistage axial flow turbomachines was initially accomplished using low-speed compressor experiments. For subsonic compressor stages higher efficiency and pressure rise were observed with decreased axial spacing. This trend is not observed in transonic rotors due to important blade-row interactions present in high speed rotors. In subsonic compressors, wake recovery [1, 5–9] has been shown to be the mechanism responsible for increased efficiency at reduced rotor/stator axial spacing. Although wake recovery could theoretically reduce wake mixing loss for transonic compressor stages, there are additional interactions that occur between blade rows, such as wake-shock and shock-vortex interactions. It is because of such transonic unsteady interactions that as axial spacing is reduced other major sources of loss are incurred. These unforeseen and unquantified blade-row interactions result in performance much lower than design expectations for some transonic compressors. An understanding of these high speed blade-row interactions will help designers more accurately predict true compressor performance in the transonic regime.

Prior experimental and computational transonic blade-row interaction research has discovered how the interaction of a rotor bow shock with an upstream stator trailing edge generates additional loss. Compressor rig tests [6, 10, 11] have shown that when stator-rotor axial spacing is reduced the pressure ratio, mass flow rate, and efficiency all decrease. Further analysis [12–14] has shown that shed vortices increase in size and strength with rotor bow shock strength, leading to more incurred loss with decreased axial gap. The formation of the vortex was found to correlate with the rotor bow shock passing. Vortices were shed from the trailing edge of the upstream stator in response to the unsteady, discontinuous pressure field generated by the impingement of the downstream rotor bow shock.

Sanders and Fleeter [15] performed PIV measurements on the wake shed from an IGV in a transonic compressor. Their results demonstrated oblique shock effects in the upstream blade row, vortex shedding, and boundary layer separation, and they concluded that the unsteadiness in the upstream blade row was driven by the downstream rotor. An oscillation in the trailing
edge stagnation point was caused by the impacting rotor bow shock, and this caused significant changes in the wake behavior.

The effects of blade loading have been investigated with regards to vortex shedding as well. Increased stator loading leads to a greater pressure rise, but at high speeds blade-row interactions can cause additional losses at increased loading. Increased incidence increases the loading on the blade, and causes a greater acceleration and subsequent deceleration of the flow on the suction side, possibly leading to flow separation farther upstream than at lower incidences. In the time-average this can cause a thicker wake to be shed, and in the time-accurate stronger vortices can be shed as a result of the higher loading. Previous experimental research has shown that increased stator loading increases the size and strength of shed vortices [16].

1.1 Motivation

This research continues to investigate the effects of axial spacing and stator loading on shed vortices in greater detail, and is part of a high-fidelity experimental and computational research effort to discover and understand the flow physics of transonic blade-row interactions and how they affect compressor aerodynamic performance. This research illustrates what happens to the strength of shed vortices in highly loaded blade rows, as well as the mechanisms governing vortex shedding characteristics.

The Blade Row Interaction (BRI) rig simulated an embedded transonic fan stage [16–20] that was designed to not only vary the stator-to-rotor axial spacing but the stator loading as well. Both experimental and computational methods have been used to analyze the BRI rig. Performance and PIV measurements were obtained for the BRI rig at numerous configurations of varying axial gap and stator loading [16,17]. It was observed that the rotor bow shock caused additional losses as the axial gap between the upstream stator and the rotor was reduced. Not only do blade-row interactions generate loss between a stator and transonic rotor, the size, strength, and trajectory of shed vortices through the rotor passage has been shown to affect rotor pressure ratio and efficiency [18–21]. Whether the mean vortex path convects near the blade surface or through the center of the rotor blade passage is important.

Previous experimental work on the BRI rig sought to determine the effects of upstream stator loading and rotor bow shock strength on the strength and size of shed vortices through
experimental PIV work [16]. It was observed that vortex shedding was synchronized to the passing of the rotor bow shock, with two strong vortices shed due to the passing shock, followed by a smaller shed vortex due to natural shedding effects. The data showed that the circulation of a vortex increased by 19 to 23% from far to mid and from mid to close spacing due to the increased strength of the rotor bow shock impacting the stator trailing edge. A reduction in upstream stator loading caused a decrease in shed vortex circulation for the same stator/rotor axial spacing by 20 to 25%. With larger and stronger vortices, more flow blockage and entropy generation is observed, which causes increased stage losses.

This research continues to analyze the BRI rig with high-fidelity, time-accurate CFD simulations. These simulations are performed at three different stator loadings and three stator/rotor axial spacings to allow a more thorough analysis of the effects of stator loading and shock strength on vortex shedding initiated by the interaction of the rotor bow shock with the stator trailing edge. In this analysis a model is developed to predict shed vortex characteristics to give compressor designers another tool to account for unsteady effects in high speed compressors.

Another focus of this research is presenting the complex flow field that exists in the stator passage upstream of a transonic rotor. The PIV measurements cited in Ref. [16] did not capture the stator suction surface flow field. Understanding the challenges and consequences of stator loading variations, thick boundary layers, and possible separation on the suction surface has not previously been investigated and will be shown to be of great importance. This research analyzes the entire unsteady stator flow field to learn the origin and cause of a low-velocity region that originates upstream of mid-chord on the stator suction surface, which has the effect of increasing shed vortex strength.

An understanding of the unsteady phenomena associated with blade loading and axial spacing in turbomachinery will help compressor designers account for unsteady flow physics at design and off-design operating conditions. Different stator loadings model off-design operating conditions for the stator row. The performance of high-speed, highly loaded turbomachines is very sensitive to operating conditions and off-design performance is very difficult to model and predict. The results of this analysis are intended to present new insights into vortex shed-
ding and stator suction side boundary layer behavior in transonic axial compressors that will allow designers to account for these unsteady effects.

1.2 Plan of Development

A review of the relevant literature will be presented in Chapter 2, with an emphasis on blade-row interactions, vortex shedding, and previous research that led to the current study. The Blade Row Interaction (BRI) rig will be explained, including previous research performed on the BRI rig. Chapter 3 will detail the numerical setup of the CFD simulations. Details will be provided on the computational domain, gridding, initialization, grid partitioning, and simulation convergence. The time-averaged results will be given in Chapter 4, comparing the CFD results to the experimental performance results. The time-accurate results will also be presented, with details on the suction side boundary layer behavior of the stator as well as the shock-induced vortex shedding flow physics. The measured circulation of the shock-induced vortices will be given in Chapter 5. A model to predict vortex circulation will be developed from dimensional analysis and design parameters. Chapter 6 will provide a further discussion of the time-averaged, time-accurate, and circulation results. Chapter 7 will draw conclusions on the results of this analysis. Various appendices also supplement this research, and are given after.
CHAPTER 2. LITERATURE REVIEW AND BACKGROUND

A brief summary of some of the relevant literature is given in this chapter in order to provide a context for this analysis. Much research concerning blade-row interactions has been performed, and the most significant literature is reviewed here. In this section wake recovery will be explained, with some relevant literature reviewed. Transonic blade-row interactions are reviewed, followed by a brief summary of some relevant literature on vortex shedding in turbomachines. An explanation and review of the Stage-Matching Investigation (SMI) research is given, and the current Blade-Row Interaction (BRI) rig is detailed, with an explanation of the geometry and hardware, followed by the relevant research already performed on the BRI rig.

2.1 Wake Recovery

Wake recovery is an important phenomenon observed in subsonic blade-row interactions. Transonic blade-row interactions are characterized by shocks, increased vortex shedding, and especially the increased losses associated with these phenomena. In modern compressors the axial gap between blade rows has traditionally been reduced to decrease engine weight. In low-speed compressors, it was found that this also led to increased efficiency. Smith [5, 7] described the concept of wake recovery, which explains the improvement in propulsive efficiency of a turbomachine when wake fluid is ingested into the propulsive stream. A portion of the drag loss associated with immersed bodies, such as stator blades or guide vanes, is offset by this improvement in propulsive efficiency as long as the wake is ingested into the propulsor (the rotor), which can likely improve the system efficiency.

This phenomenon occurs due to the propulsor having a natural tendency to add more energy to fluid at lower velocity, which when applied to a wake defect in the axial velocity flow field, causes the velocity defect to decrease and the wake to flatten as seen in Figure 2.1. As the wake passes through the propulsor in Figure 2.1 the low-momentum fluid in the wake region
is accelerated more than the freestream fluid, which causes the wake deficit to decrease at the propulsor exit. This flattening of the wake is called wake recovery, since some of the velocity defect is recovered. Wake recovery is most efficient when the propulsor is positioned to ingest a wake before the wake has had time to dissipate. This phenomenon is thought to partially explain why there is an increase in efficiency in turbomachines with reduced axial spacing in experimental rigs.

Smith found that wake recovery increased with wake defect, thus, from a design standpoint, if a strong wake defect is encountered in the flow it should be directed into the downstream propulsor in order to recover some of the drag loss. Also, wake recovery in high thrust-loading propulsors, such as turbomachinery, was observed to be the greatest.

Deregal and Tan [8] performed unsteady two-dimensional numerical simulations to investigate possible benefits in compressor performance when rotor wakes are mixed out after the stator blade passage as opposed to before. The motivation for this was Smith's observation that an increase in compressor performance occurs when the axial gap is reduced. Specifically the investigation sought to determine the time-averaged overall static pressure rise and mixing loss when the rotor wakes are mixed out before and after the stator passage. It was shown that the pressure rise and the loss associated with mixing scale quadratically with the velocity defect in the wake and linearly with its width. Numerical results clearly show that mixing losses are reduced if mixing occurs after the stator passage as opposed to before. This allows wakes to
be stretched in the stator passage, decreasing the velocity defect in the wakes near the passage outlet compared to the inlet, which is the result of wake recovery.

Research by van Zante [9] showed that wake recovery is an important mechanism in decreasing subsonic compressor loss. Wakes shed from upstream blade rows in an axial compressor decayed as they passed downstream due to mixing (resulting in loss) and stretching (resulting in no loss). Wake recovery was explained as wake decay by inviscid stretching with no loss accrual as shown previously by Smith [5]. For axial spacing typical in core compressors, where axial spacing has been significantly reduced, wake stretching was shown to be the dominant wake decay process within the stator. A design tool to model wake recovery was provided, but it was limited to two-dimensional, incompressible flow assumptions, while neglecting radial transport and wake migration. It was noted that the model underpredicted wake stretching compared to experimental data, and that other viscous losses in the stator passage were not modeled.

Van Zante also predicted that higher stage loading may be possible if wake recovery is utilized to its full advantage. Rotor/stator clocking affects the mean wake path through subsequent blade passages. Wake recovery may be maximized by optimizing the clocking for a given stage. This could cause a substantial percentage of the rotor wake decay to be recovered in the downstream stator passage, however, it was noted that other mixing losses could be increased due to increased stage loading.

2.2 Blade-Row Interactions

Although this section describes some subsonic rotor/stator interactions as well as transonic stator/rotor interactions, many similarities exist. Propagating wakes, shocks, and vortices, and the interactions of these flow phenomena are typical features of blade-row interactions.

Although wake recovery was shown to lead to an increase in stage efficiency for low-speed compressors, this trend was not observed for transonic rotors, where significant other blade-row interactions were present. The interaction of a shock with a blade, boundary layer, wake, or vortex is shown in this section to be very important, as additional losses are incurred.
Wake recovery is still present, but the others losses present at high speeds overpower any propulsive wake recovery benefit.

An experimental investigation by McCormick [22] of cambered flat-plates designed to simulate highly-loaded airfoils in a cascade test facility showed increased boundary layer thickness at higher loading. An intermittent boundary layer separation location was observed, indicative of vortex shedding. The mean suction side separation location moved farther upstream at increased loading, and also resulted in a larger trailing edge interaction region. This caused the pressure side flow deviation to increase significantly more at higher loading, resulting in less airfoil circulation than was predicted due to extra losses.

Work by Epstein [23] showed the importance of accounting for vortex streets in compressor blade wakes, as wakes are not merely turbulent velocity defects. Experimental results showed that vortex shedding was an important aspect of blade wakes, especially regarding the time-mean flow measurements, and numerical results suggested that the shedding process was more complex than can be represented by a simple Strouhal number. Phase-locking was suggested to be an important consideration in trailing edge vortex formation. This has been proven by further work as shown below.

Numerical simulations performed and analyzed by Valkov [24] showed the effects of wake propagation through a stator passage and their effect on upstream suction surface vortex formation. As the passing rotor wake migrated downstream the low-momentum boundary layer fluid on the suction surface was convected away from the wall, forming vortices. These vortices changed the boundary layer behavior on the stator downstream. It was also found that the shed vortex strength was affected by the foremost suction surface pressure gradient.

Numerical work by Valkov and Tan [25, 26] investigating the effects of upstream rotor wakes on stator performance further verified Smith's research [5], and showed that wake recovery at subsonic rotor speeds leads to decreased losses in compressor stages. Passage loss was found to increase with inlet wake velocity defect, but in the range of wake defects analyzed the benefits of wake recovery were greater than the passage loss increase. This phenomenon occurs because recovery occurs over a shorter length scale than turbulent diffusion. The largest passage loss was encountered on the stator suction side due to the boundary layer distortion of the wake “negative jet” on the stator suction side, which can be seen in Figure 2.2. The “negative jet"
effect is a phenomenon that is observed as the freestream fluid velocity is subtracted from the flow field. This results in wakes appearing to act as negative jets, as shown in Figure 2.2 which shows the passing rotor wake acting as a negative jet on the downstream stator suction surface. This wake negative jet causes the lower total pressure fluid in the boundary layer to convect away from the wall and mix with the high-momentum freestream fluid, causing a larger time-averaged passage loss.

It was also shown that although vortical structures are inherently three-dimensional, they can be described in two-dimensional terms similar to wake recovery. Most of the tip leakage vortex energy is recovered through the stator, but there is a significant increase in passage loss in the tip region. The amount of recovery from the tip leakage vortex is significant, but can be offset by the loss accrued in the tip region. Viscous simulations showed that the “negative jet” effect on the suction side of the stator is stronger than in the inviscid simulations due to the lower momentum boundary layer fluid being convected farther from the wall region. This leads to a higher passage loss for viscous simulations, as the high entropy fluid is “lifted” from the suction side of the stator. This helped to illustrate the importance of including viscous effects where blade-row interactions are important.
Experimental work by Sanders and Fleeter [15] showed that the trailing edge stagnation point on an upstream stator is highly affected by a passing rotor bow shock. As the rotor bow shock impacted the upstream stator trailing edge the shock caused an increase in the static pressure on the pressure side, which caused the pressure gradient at the trailing edge to be modified. It was observed that the rotor bow shock interaction with the stator trailing edge caused the stagnation point to oscillate back and forth from the pressure and suction surfaces due to this trailing edge pressure gradient. It was this oscillation that caused vortex shedding to occur at the blade passing frequency.

Experimental work by Sanders [27] showed that rotor wakes passing through downstream stators were chopped by the stator leading edge, and as they propagated through the stator passage the “negative jet” effect caused the low momentum fluid to convect from the suction side to the pressure side of the passage. This caused the wake near the pressure side to thicken, and the suction side to thin. Experiments were performed with low and high speed rotors, with the high speed transonic rotor wakes dissipating much more rapidly than the respective low speed rotor wakes. This was surprising since the transonic rotor wakes were much larger and stronger than the low speed rotor wakes. The wakes dissipated more rapidly because of unsteady flow effects, specifically because the convection of the wake from the suction side to the pressure side of the stator blade passage was more pronounced for the larger wakes, causing the wake effects to disappear on the suction side and in the freestream. At the trailing edge, the stagnation point oscillated back and forth due to the impacting rotor bow shock.

Ottavy [28] performed measurements and analysis on the interaction between the rotor bow shock and the wake shed from an inlet guide vane. The experiment used laser two-focus anemometer measurements between an IGV and transonic rotor. The results showed that the shock wave had a large effect on the wake. Upstream of the shock wave the wake depth was reduced, and the wake was overturned as a result of an expansion zone in the flow due to the curvature of the rotor blade suction surface. Downstream of the shock wave the wake-deficit increased, and the wake was underturned.

Mailach and Vogeler [29] presented experimental results of the first stage of a four-stage subsonic compressor. The stator boundary layer on both the pressure and suction side were shown to be highly affected by the rotor wake impingement on the boundary layers. On the
pressure side it was shown that the boundary layer was in a transitional state along the entire chord length, however, the suction side of the stator exhibited a periodic shifting of a wide transitional boundary layer upstream and downstream with the passing of the rotor wake. The boundary layer fore of midchord was laminar in between wakes, and as the wake began to impinge on the suction side, causing a “negative jet” effect, the boundary layer changed to a transitional state. This region propagated downstream causing a fully-turbulent section to form, followed by a boundary layer transition back to laminar after the rotor wake-passing event. This periodically formed with the rotor wake passing, causing additional loss.

Further work by Mailach and Vogeler [30] showed that the unsteady pressure distribution on the stator blades was affected by both the upstream wake impingement and the downstream potential flow field, but was independent of the wake propagation through the blade passage. At the design point there was a strong acceleration on the first 25% chord of the suction side, followed by a strong deceleration and adverse pressure gradient. This caused an increase in the pressure RMS fluctuations, especially in the region of the strongest adverse pressure gradient. Both the pressure and suction side had double peaks in the pressure fluctuations for each blade passing. Each of the peaks were associated with one of two causes: (1) the upstream wake from the passing rotor, or (2) the change in the downstream potential flowfield due to the passing of the downstream rotor, which propagated upstream at $M = 1$ from the trailing edge to the leading edge of the stator. The double peaks were periodic since these blade rows all contained the same number of blades per row.

Experimental analysis by Soranna [31] showed the importance of the impingement of an inlet guide vane wake on a downstream rotor boundary layer and near-wake region. The focus of the analysis was on the suction side boundary layer near the rotor trailing edge, and on the pressure and suction side near wake regions. It was observed that when the upstream wake impinged on the rotor blade the suction side boundary layer thinned significantly and momentum thickness decreased. Investigation of the governing equations revealed that the phase-averaged unsteady term was the main cause of the decrease in momentum thickness. It was also observed that the boundary layer thinning extended into the near wake region, making it narrower, thus increasing the shear velocity gradients and turbulent kinetic energy and production rate. The location of the wake impingement on the rotor also affected the velocity
field outside the boundary layer. When the wake impinged further upstream on the rotor the rotor wake acted as a negative jet, causing large shear velocity gradients. When the upstream wake impinged farther downstream the rotor wake turned gradually more in the mean flow direction, decreasing the shear velocity gradients. This caused the fluid in the regions just outside the boundary layer to behave in a similar manner.

From the previous literature reviewed it has been shown that blade-row interactions have a great impact on rotor and stator performance. Separation increases at increased stator loadings, and thicker boundary layers result downstream. Wake impingement from upstream blade rows affects downstream blade row boundary layer behavior, and subsequent losses at both subsonic and transonic speeds. In high speed compressors the rotor bow shock provides a much more complex flow field, and additional losses are incurred as the shock interacts with the upstream vanes, wakes, and vortices. The passing rotor bow shock causes the stagnation pressure location to oscillate on the trailing edge. This affects vortex shedding in transonic compressors. The negative jet effect causes drastic changes to boundary layers. The boundary layer behavior of a blade, wake migration, and vortex shedding all affect the pressure distribution on blades, and can cause additional performance loss.

2.3 Vortex Shedding

Vortex shedding has been shown to be an important phenomenon in compressors. Strong vortices can cause radial transport of high-entropy, low-momentum fluid from the tip clearance flows to spiral radially inward, causing additional losses downstream. Strong vortices have strong shear layers, and generate more entropy as they dissipate than weaker vortices. Large vortices can also act to block the flow, and cause streamlines to deviate from axial, also causing additional losses. A propagating vortex can also interact with shocks and boundary layers, which can cause additional losses.

Detailed trailing edge pressure profiles in a turbine cascade were obtained by Cicatelli [32] in order to investigate vortex shedding in turbomachines. It was observed that the trailing edge pressure oscillated at the vortex shedding frequency, though a dominant vortex shedding frequency was not observed at the low Mach number present in the flow. Phase-averaged trailing edge pressure distributions showed that pressure variations were slightly higher on the
pressure side, most likely due to the circulation of the vane. The vortex shedding mechanism was also explained. A vortex formed due to the circulation from the upstream shear layer until the vortex was strong enough to entrain fluid from the opposite shear layer. As this region of opposite circulation grew, it cut off the supply of circulation to the growing vortex, and the vortex was shed.

Numerical simulations performed by Currie [33] showed the importance of resolving vortex shedding in computational efforts. Quasi-three-dimensional unsteady simulations were performed for a transonic turbine cascade in order to study trailing edge vortex shedding. Turbine blades are characterized by thick, blunt trailing edges, causing vortex shedding to be a significant source of unsteady losses in the near wake region. Results showed that a decrease in vortex shedding intensity at increased Mach number caused the pressure losses to decrease.

A combined experimental and numerical effort by Gottlich [34] showed that vortex shedding in turbine vanes was phase-locked to the passing of the downstream rotor. The blade-shock interaction is initiated in a slightly different manner for turbines than for transonic compressors, but the physics are similar. The vane suction side shock extends downstream into the rotor passage, where it is reflected off the passing rotor back upstream. As the reflected pressure wave interacts with the vane trailing edge it induces vortex separation. These results showed a strong correlation between the rotor passing frequency and the vortex shedding. Interestingly, the pressure side vortices were observed to be stronger than the suction side vortices due to the thinner boundary layer and steeper velocity gradient normal to the vane. A modulation in the vortex strength within a rotor passing period was also observed. This is thought to occur due to the change in the vane suction side shock strength over a rotor passing period.

Two-dimensional numerical simulations performed by Zachcial and Nurnberger [35] showed that blade-row interactions and vortex shedding in turbomachines are closely related. It was observed that the vortex shedding was phase-locked to the rotor blade passing frequency, and that the interaction of the rotor bow shock with the upstream stator boundary layer induces separation. This interaction produces a very different wake structure than at subsonic conditions as it contains various shed vortices that are triggered by the periodic nature of the passing rotor bow shock. These vortices were shown to affect downstream rotor performance, especially with varying axial gap, by affecting the rotor boundary layer development.
An embedded transonic stator row was modeled and studied in a linear cascade by Langford [14] to determine the effects of a shock wave impingement on a compressor stator trailing edge. A shock tube provided a moving shock of three different strengths propagating upstream. Figure 2.3 shows a sequence of shadowgraph images showing the shock-induced vortex formation at the trailing edge of the stator. The shock-induced vortex forms at the trailing edge in image 6, then propagates downstream in images 9 and 12. Similar to observations made by Gorrell [11, 12], the shock turned more normal as it propagated up the stator passage, resulting in additional losses. It was also found that a vortex formed on the stator trailing edge as the shock passed, and the shed vortex size and strength was directly related to the incoming shock strength by Equation 2.1, where \( dV \) is the velocity deceleration across the shock and \( \beta \) is the angle of the shock with respect to the tangential direction.

\[
\Gamma = 0.2 C_x dV \cos \beta
\]  

(2.1)

Research by Turner [36] showed the importance of aerodynamic blockage on engine performance. Blockage is a measure of three-dimensional effects. It demonstrates the location of shocks, secondary flows, separated regions, large wakes, and blade-row interaction effects, which are all sources of loss in turbomachinery. Langford found that blockage due to the shock-induced vortex on the trailing edge was also found to increase with shock strength. This showed that a decrease in compressor performance would be experienced as these three-dimensional effects become more present in the flow.

As a follow-up to Langford’s work, van de Wall [37] utilized two-dimensional numerical simulations to investigate the unsteady flow mechanisms present in a rotor bow shock impinging on a stator trailing edge. The CFD results compared well with the experimental results, as shown in Figure 2.4. The refraction and reflection of the rotor bow shock in the stator passage was shown, as well as the vortex formation at the trailing edge and propagation downstream at an increased deviation. It was observed that the vortex caused a significant blockage, which in turn caused a large suction side separation at about 30% chord from the trailing edge. A comparison to PIV data, shown in Figure 2.5, shows the shed vortex and the separation region due to the associated blockage. Due to the success of these numerical simulations in accu-
rately resolving shock-induced vortex formation, the recommendations given for performing time-accurate numerical simulations where vortex shedding is important were followed in this research as much as possible.

2.4 Stage Matching Investigation (SMI) Rig

The Stage Matching Investigation (SMI) rig, built at AFRL, was an experimental test rig from which the rig used in this analysis was derived. The SMI rig used an uncambered, blunt trailing edge wake generator (WG) upstream of a transonic rotor, where the axial spacing between the two blade rows could be changed from close to mid to far spacing. The wake generator caused a thick wake to be shed due to base drag, rather than viscous diffusion. The number of wake generators was variable, so multiple configurations were tested (12 WG or 24 WG). A midspan section showing close and far axial spacing is shown in figure 2.6. Various experimen-
tal and computational analyses were performed for this geometry in order to better understand blade-row interactions.

Experimental analysis of the SMI rig by Gorrell [11] showed a decrease in compressor stage performance with axial spacing due to blade-row interactions not seen in low-speed compressors. Figure 2.7 shows that when axial spacing was reduced pressure ratio, mass flow rate, and efficiency all decreased. This trend was most evident at design speed, and not at lower speeds. Further numerical work [12] on the same test rig was done to investigate the effects
of axial spacing on compressor performance more fully. It was found that the rotor bow shock interacted with the upstream wake generator. At close spacing, the strong rotor bow shock was chopped by the wake generator trailing edge, causing the oblique shock to turn more normal, thereby resulting in greater loss as the pressure wave travelled upstream. At far spacing the rotor bow shock degenerated to a bow wave, and the weaker bow wave was blocked by the wake generator trailing edge, resulting in negligible losses associated with the interaction. This resulted in the highest efficiency and pressure ratio at far spacing.

Simulations of the SMI rig at close and far spacing using time-accurate CFD and an Average Passage approach were performed by Turner [38] showing the three-dimensional nature of the flow and the radial migration of wakes and shed vortices. The unsteady results com-
pared well with the experimental data, showing that at 90% span the close spacing configuration was more efficient than far, but below 80% span far was more efficient than close. This efficiency profile crossover was attributed to the radial migration of upstream shed vortices. Entropy was high near the casing due to endwall and clearance flows, and this region of high entropy convected to lower spanwise locations due to a negative radial velocity near the casing in the vicinity of the vortex. A vortex core just downstream of the shock, where the axial velocity had decreased, showed the radial velocity was negative only near the casing, and this caused the high entropy fluid near the casing to migrate radially inward in order to conserve angular momentum. Fluid particles with a positive radial vorticity in the vicinity of the vortex spiraled radially inward carrying high entropy fluid toward midspan. This effect was amplified at close
axial spacing as the stronger rotor bow shock caused a stronger axial deceleration, resulting in more radial transport.

Further experimental PIV work and numerical simulations on the SMI rig by Gorrell [13] showed that the vortex shedding from the wake generator was phase-locked to the rotor passing. Close and far spacing were examined, and it was found that at close spacing vortices were shed as a result of the unsteady, discontinuous pressure field on the wake generator trailing edge caused by the passing rotor bow shock. The size and strength of the shed vortices was shown to increase with decreased axial spacing. A model to predict the change in maximum vorticity in a vortex was also developed, and is given in Equation 2.2, where $\phi$ is the shock angle with respect to the blade angle, $\Delta p$ is the change in static pressure across the shock, and $\bar{p}$ is the average pressure across the shock.

$$\Delta \omega = \left( \frac{4}{\pi \tan \phi} \right) \left( \frac{\Delta p}{\bar{p}} \right) \left( \frac{\bar{p}}{m} \right) \left( \frac{S}{t} \right)$$ (2.2)

Further numerical work on the SMI rig by Nolan [21, 39] investigated wake-phasing in conjunction with blade-row interactions. The impact of wake-phasing on performance, both time-averaged and unsteady, were examined by changing the vortex trajectory and stagnation pressure nonuniformity through the downstream rotor passage. Since the vortex shedding was phase-locked to the rotor position there was a time-averaged, pitchwise nonuniformity entering the rotor passage, which was found to affect rotor performance. When the rotor was surrounded by high stagnation pressure fluid (phase-locked such that the vortex pair trajectory was along the blade), the boundary layer thickness, displacement thickness, and deviation were decreased leading to a very large increase in pressure rise, flow turning, and thus, work input by 4-5%.

### 2.5 Blade-Row Interaction (BRI) Rig

The Blade Row Interaction (BRI) rig, shown in Figure 2.8 was a variation of the AFRL SMI rig [10, 11, 13]. The current work utilizes the BRI rig geometry to study the effects of stator loading on vortex shedding. The BRI rig used much of the same hardware as the SMI with the main difference being the replacement of the blunt, uncambered wake generator of the SMI rig.
2.5.1 BRI Geometry & Hardware

The rotors used in the SMI and BRI rigs were designed for axial inlet flow and, thus, required a swirler and deswirler to maintain axial inlet flow to the rotor, as shown in Figure 2.9.
These rows generated a wake through viscous effects rather than base drag, and the deswirler had a more realistic stator trailing edge geometry. As the axial inlet flow passed through the swirler row, a tangential velocity (30 degrees of turning) was introduced to the flow. The deswirler was a highly loaded stator with a design diffusion factor of 0.45. The design intent was to move the loading as far forward as possible without leading edge separation. Stator loading was changed by adjusting the stagger angle of the swirler, which changed the incidence to the deswirler, and consequently the suction side boundary layer thickness. This is displayed in Figure 2.10, which shows the swirler and deswirler rows with the swirler angle varied to yield decreased and nominal loading on the deswirler. Additionally, the swirler row could be clocked relative to the deswirler to control the pitchwise position of the wake from the first blade row and to optimize the total pressure loss at the entrance of the rotor. A stator row downstream of the rotor was also present. The rig was designed to permit the stator-to-rotor axial spacing to be set to three values—“close,” “mid,” and “far”—as shown in Figure 2.8. The average non-dimensional distances between the deswirler trailing edge and the rotor leading edge at the casing are given in terms of the axial distance divided by the mean chord of the deswirler: close spacing 0.23, mid spacing 0.48, and far spacing 1.0.

Figure 2.10: Midspan Slices of the Swirler and Deswirler Rows for Decreased and Nominal loading
The design parameters of the BRI rig stage are summarized in Table 2.1. The rotor and stators in the BRI rig are different from those used in the SMI rig. Thus, direct performance comparisons between the SMI and BRI rigs should be done with caution. The SMI simulated an embedded transonic core stage, while the BRI rig simulates an embedded fan stage. The major differences between the fan and core stages are fewer rotor blades (28 in the fan versus 33 in the core) and higher tip speed (414.53 m/s in the fan versus 341.37 m/s in the core), resulting in the tip relative Mach number being increased (1.389 versus 1.191) and the hub relative Mach number being transonic (1.100 versus 0.963).

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Deswirler</th>
<th>Rotor</th>
<th>Stator</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of airfoils</td>
<td>32</td>
<td>28</td>
<td>49</td>
</tr>
<tr>
<td>Average aspect ratio</td>
<td>1.24</td>
<td>0.916</td>
<td>0.824</td>
</tr>
<tr>
<td>Flow/annulus area (((\text{kg/s})/m^2))</td>
<td>—</td>
<td>196.30</td>
<td>—</td>
</tr>
<tr>
<td>Corrected tip speed (m/s)</td>
<td>—</td>
<td>414.53</td>
<td>—</td>
</tr>
<tr>
<td>(M_{\text{Rel}}) hub</td>
<td>0.750</td>
<td>1.100</td>
<td>0.830</td>
</tr>
<tr>
<td>(M_{\text{Rel}}) tip</td>
<td>0.720</td>
<td>1.389</td>
<td>0.700</td>
</tr>
<tr>
<td>Max D Factor</td>
<td>0.45</td>
<td>0.545</td>
<td>0.506</td>
</tr>
<tr>
<td>LE tip diameter (m)</td>
<td>0.4825</td>
<td>0.4825</td>
<td>0.4825</td>
</tr>
</tbody>
</table>

2.5.2 BRI Analyses

Experimental work on the BRI rig by Estevadeordal [17] showed that the deswirler wake was flatter for close spacing than at far spacing at peak efficiency. The deswirler wake thickness also decreased with decreased stator loading (lower incidence). Vortex shedding was observed to be phase-locked to the rotor bow shock. Interestingly, different vortex shapes were observed for the same configurations at different operating conditions. These observations were made at peak efficiency and near stall.

Numerical simulations performed by List [18–20] showed the effects of blade-row interactions in the BRI rig at axial spacings of close, mid, and far for a single blade loading. It was
shown that vortex shedding was phase-locked with the rotor, and the rotor bow shock passing caused a large vortex to be shed. Performance results were different than those for the SMI rig. The highest efficiency was found to occur at mid spacing, with far spacing having slightly lower efficiency, and close having the lowest. The trajectory of the vortex path through the rotor passage was found to have a significant impact on performance. At mid spacing the vortex mean trajectory was through the center of the passage, while the trajectory for far and close was near the rotor blade, causing an interaction of the vortex with the rotor boundary layer to occur. This caused a decrease in stage performance for far and close. For mid and far spacing the upstream stator wakes were mostly mixed out before passing through the rotor passage.

An experimental analysis on the BRI rig was performed by Reynolds [16, 40] using PIV to investigate velocity fields in the vicinity of the deswirler trailing edge in order to investigate the effects of stator loading and rotor bow shock strength on deswirler shed vortex size and strength. It was observed that vortex shedding was synchronized to the passing of the rotor bow shock, as in other SMI and BRI investigations. At close spacing the vortex shedding appeared 'strongly forced' and 'phase-locked' to the rotor passing. The influence of the rotor bow shock on vortex shedding decreased with axial distance. This is because the rotor bow shock weakens before impacting the deswirler trailing edge at greater axial spacing. It was found that the shed vortices were non-axisymmetric, and, as can be seen in Table 2.2, the pitchwise radius of the shed vortices increased directly with shock strength (19% from far to mid, and 19% from mid to close), and changed negligibly with varying loading. Three vortices were observed for each rotor passing period, with two strong, followed by one weak. Table 2.3 shows the circulation of the strong, shock-induced vortices.

<table>
<thead>
<tr>
<th>Spacing</th>
<th>Loading</th>
<th>Radius (mm)</th>
<th>Pitchwise Radius (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>close</td>
<td>nominal</td>
<td>1.71</td>
<td>1.64</td>
</tr>
<tr>
<td>mid</td>
<td>nominal</td>
<td>1.80</td>
<td>1.33</td>
</tr>
<tr>
<td>mid</td>
<td>decreased</td>
<td>1.69</td>
<td>1.28</td>
</tr>
<tr>
<td>far</td>
<td>nominal</td>
<td>1.80</td>
<td>1.09</td>
</tr>
<tr>
<td>far</td>
<td>decreased</td>
<td>1.82</td>
<td>1.12</td>
</tr>
</tbody>
</table>
Figure 2.11 shows velocity contours and vorticity lines of the ensemble-averaged PIV results of the deswirler trailing edge at mid spacing for both decreased and nominal loading. Seven images per blade passing are shown at 20 µs increments. The rotor bow shock can be seen to interact with the trailing edge and propagate upstream. Three shed vortices were observed for each blade passing, with A and B being strong, and C being weaker than both A and B. The circulation of the strong vortices was shown to increase with stator loading from decreased to nominal (25% for mid, and 20% for far), and increase with shock strength from mid to far (23% at nominal loading, and 19% at decreased loading). The strength of the weaker vortices changed little with shock strength, but increased with increased loading. From these results it was concluded that the strong vortices were shed due to the passing of the rotor bow shock, and increased shock strength resulted in stronger shed vortices, but no design rule or model was described.

<table>
<thead>
<tr>
<th>Spacing</th>
<th>Loading</th>
<th>Circulation $\left( \frac{m^2}{s} \right)$</th>
</tr>
</thead>
<tbody>
<tr>
<td>close</td>
<td>nominal</td>
<td>0.705</td>
</tr>
<tr>
<td>mid</td>
<td>nominal</td>
<td>0.752</td>
</tr>
<tr>
<td>mid</td>
<td>decreased</td>
<td>0.568</td>
</tr>
<tr>
<td>far</td>
<td>nominal</td>
<td>0.580</td>
</tr>
<tr>
<td>far</td>
<td>decreased</td>
<td>0.462</td>
</tr>
</tbody>
</table>

### 2.6 Summary

The literature shows that compressor performance is highly dependent on unsteady flow phenomena, such as blade-row interactions. In a transonic compressor vortex shedding is strongly forced and phase-locked to the passing of the rotor bow shock at decreased axial spacing. Various studies show that stator-shed vortex strength increases with shock strength and stator loading. These strong vortices are shown to cause a large blockage in the flow, increase shear gradients and turbulence, and cause further losses as they propagate downstream through the rotor passage, interact with the rotor boundary layer, and change the pressure distribution. Wake recovery is beneficial at low speeds, but these other loss-generating blade-row
Figure 2.11: PIV Results [16] Showing Deswirler-Shed Vortices at Mid Spacing.
interactions observed at transonic speeds far overpower any performance gains observed due
to wake recovery effects. Various models predict shock-induced vortex strength due to varying
shocks, but no models exist which take into account a varying stator loading. This research
aims to fill that gap in contemporary understanding of blade-row interactions.
CHAPTER 3. NUMERICAL SETUP

This chapter describes the numerical setup of the current simulations. An overview of the grid generation, flow initialization and grid partitioning is given. Some details about the flow solver, TURBO, are given with a description of the boundary conditions. The simulation convergence is described in detail with an emphasis on achieving faster convergence for similar future simulations. A low-frequency oscillation in the mass flow rates is described and explanations are given as to how time-averaged and time-accurate data was taken in order to minimize the effects of the oscillation.

3.1 Computational Domain

For this research, periodic, quarter annulus simulations of the BRI rig were performed to adequately resolve stator/rotor blade-row interactions. The computational domain includes the first three rows of the BRI test rig. Periodic grids with 8 swirler passages, 8 deswirler passages, and 7 rotor passages were generated in order to produce the high-fidelity, time-accurate simulations, as shown in Figure 3.1. Between 142-166 million nodes were gridded in each simulation, resulting in hundreds of thousands of CPU hours per simulation running in parallel on 500-1000 processors, depending on available computing resources.

It should be noted that the fourth blade row, the downstream stator, is not modeled in this simulation, as its inclusion would have required full-annulus periodic simulations rather than quarter annulus simulations. The inclusion of the downstream stator would have resulted in grids with node counts near 800 million, and the computational resources available did not justify its inclusion. This is also justified since the purpose of this research is to investigate the effects of the rotor bow shock, stator loading, and vortex shedding from the upstream stator row (deswirler), and the downstream stator has a minimal, if any, effect on this.
The numerical simulations were performed for close, mid, and far spacing configuration with three stagger angles of $-3.0^\circ$, $0.0^\circ$, and $+1.5^\circ$, which correspond to decreased, nominal, and increased loading respectively. Mid spacing is analyzed in detail in this research, while only representative results are given for far and close spacing.

### 3.2 Grid Generation

The Turbomachinery Gridding System [41] (TGS) was used for meshing the computational domain. TGS creates block hexahedral H-meshes, with elliptical smoothing at the leading and trailing edges to reduce the generation of high aspect ratio cells at the blade inlet and exit planes that are associated with block meshes. This results in a fine grid that is uniformly distributed across the flow domain, resulting in better solver stability. Grid density increases with each downstream blade row in order to capture unsteady flow features as they propagate downstream. Figure 3.2 shows two blade passages of each blade row with every fourth node shown. Special consideration was given to the region between the deswirler trailing edge and rotor leading edge in order to adequately resolve rotor bow shocks, shed vortices, and the unsteady interactions of the deswirler and rotor. Details of the grid are shown in Figures 3.3
Figure 3.2: Grid at Midspan Showing Two Blade Passages of Each Blade Row (Every 4th Node Shown). Coordinate Axis Values are in Meters.

and 3.4. Figure 3.3 shows the region between the deswirler trailing edge and rotor leading edge at mid spacing. Figure 3.4 shows details of the deswirler trailing edge. Over 50 cells are placed on the trailing edge in order to capture vortex shedding effects.

The spatial and temporal resolutions for the simulations match those of List, who performed a grid and time step independence study of this geometry previously [19]. The final node counts are given in Table 3.1 for each blade row. It should be noted that the number of tangential cells increased from 150 in the swirler, to 200 in the deswirler, to 225 in the rotor passage. A constant 100 cells in the radial direction was used for all three blade passages. The final $y^+$ values ranged from just under one up to three. In order to adequately resolve flow features, especially vortex shedding, the node counts match or exceed the recommendations of van de Wall [37], which include nearly 400 nodes in the axial direction and 160 nodes in the tangential direction. According to van de Wall, temporal resolution is also important in order to properly resolve vortex formation and propagation through the rotor passage. Although 800 time steps per rotor blade passing is recommended, List [19] found that 320 time steps per blade passing provided a reasonable trade-off between resources and accuracy, and was sufficient to resolve
Figure 3.3: Grid Details at Midspan Between the Deswirler and Rotor. Coordinate Axis Values are in Meters.

flow features with these grids. Work by van de Wall showed that the grid was the most important factor in resolving vortex shedding, thus very fine grids are used for this study.

The rotor tip clearance region was resolved using eight radial cells in the clearance gap, allowing for adequate resolution of tip flows as suggested by Van Zante [42]. In the experimental rig the swirler blades were mounted on a button to allow for multiple stagger angles without changing out rig hardware. This resulted in a clearance gap at both the hub and tip. The hub and tip clearance gaps for the swirler row were modeled with four and eight radial cells respectively. The button was taken into account in the simulations in order to allow for greater flow accuracy in the hub and tip regions. This was done by manually inserting the button location into the simulation by manipulating the boundary conditions in the hub and tip clearance regions. More details on this can be found in reference [19].
3.3 Initialization and Grid Partitioning

The flow was initialized by interpolating the total pressure, total temperature, static pressure, and flow angle across each blade in both the axial and the radial directions. This was done because a good initial solution has a better chance of getting through the initial transients, and can result in faster convergence. The values used for each blade are given in Table 3.2, and are normalized by reference values of $p_{ref} = 101.325$ kPa and $T_{ref} = 288.15$ K. A custom code was used to linearly interpolate the pressure and temperature values at the hub and case and leading and trailing edges across the blade passage. For example, in the hub region of the swirler, $p_s$ was varied linearly from the leading edge value of 0.764 to the trailing edge value of 0.729. At the casing $p_s$ varied from 0.754 to 0.747 from the leading edge to the trailing edge.

Figure 3.4: Grid Details of Deswirler TE at Midspan. Coordinate Axis Values are in Meters.
Table 3.1: Node Counts for Each Blade Passage in the Current Numerical Simulations.

<table>
<thead>
<tr>
<th></th>
<th>Close</th>
<th>Mid</th>
<th>Far</th>
</tr>
</thead>
<tbody>
<tr>
<td>(8×) Swirler</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Axial</td>
<td>323</td>
<td>293</td>
<td>263</td>
</tr>
<tr>
<td>Radial</td>
<td>101</td>
<td>101</td>
<td>101</td>
</tr>
<tr>
<td>Tangential</td>
<td>151</td>
<td>151</td>
<td>151</td>
</tr>
<tr>
<td>Total</td>
<td>4.9 mil</td>
<td>4.5 mil</td>
<td>4.0 mil</td>
</tr>
<tr>
<td>(8×) Deswirler</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Axial</td>
<td>256</td>
<td>361</td>
<td>461</td>
</tr>
<tr>
<td>Radial</td>
<td>101</td>
<td>101</td>
<td>101</td>
</tr>
<tr>
<td>Tangential</td>
<td>201</td>
<td>201</td>
<td>201</td>
</tr>
<tr>
<td>Total</td>
<td>5.2 mil</td>
<td>7.3 mil</td>
<td>9.4 mil</td>
</tr>
<tr>
<td>(7×) Rotor</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Axial</td>
<td>391</td>
<td>391</td>
<td>391</td>
</tr>
<tr>
<td>Radial</td>
<td>101</td>
<td>101</td>
<td>101</td>
</tr>
<tr>
<td>Tangential</td>
<td>226</td>
<td>226</td>
<td>226</td>
</tr>
<tr>
<td>Total</td>
<td>8.9 mil</td>
<td>8.9 mil</td>
<td>8.9 mil</td>
</tr>
<tr>
<td>TOTAL</td>
<td>142 mil</td>
<td>154 mil</td>
<td>166 mil</td>
</tr>
</tbody>
</table>

Radial interpolation also was used from the hub to the casing with the values give in Table 3.2. Using values of $R = 287.567 \text{ J/(kg*K)}$ and $\gamma = 1.41029$, and the pressure and temperature values given in Table 3.2, velocities were calculated and the entire computational domain was initialized.

Table 3.2: Normalized Values of Initialized Flow Field. ($p_{ref} = 101.325 \text{ kPa}, T_{ref} = 288.15 \text{ K}$)

<table>
<thead>
<tr>
<th>Location</th>
<th>$p_t$</th>
<th>$p_s$</th>
<th>$T_t$</th>
<th>$\alpha$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Swirler</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Hub LE</td>
<td>1.000</td>
<td>0.764</td>
<td>1.000</td>
<td>0</td>
</tr>
<tr>
<td>Case LE</td>
<td>1.000</td>
<td>0.754</td>
<td>1.000</td>
<td>0</td>
</tr>
<tr>
<td>Hub TE</td>
<td>0.983</td>
<td>0.729</td>
<td>1.000</td>
<td>-19</td>
</tr>
<tr>
<td>Case TE</td>
<td>0.982</td>
<td>0.747</td>
<td>1.000</td>
<td>-19</td>
</tr>
<tr>
<td>Deswirler</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Hub LE</td>
<td>0.983</td>
<td>0.729</td>
<td>1.000</td>
<td>-19</td>
</tr>
<tr>
<td>Case LE</td>
<td>0.982</td>
<td>0.747</td>
<td>1.000</td>
<td>-19</td>
</tr>
<tr>
<td>Hub TE</td>
<td>0.966</td>
<td>0.764</td>
<td>1.000</td>
<td>0</td>
</tr>
<tr>
<td>Case TE</td>
<td>0.966</td>
<td>0.754</td>
<td>1.000</td>
<td>0</td>
</tr>
<tr>
<td>Rotor</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Hub LE</td>
<td>0.966</td>
<td>0.774</td>
<td>1.000</td>
<td>0</td>
</tr>
<tr>
<td>Case LE</td>
<td>0.966</td>
<td>0.757</td>
<td>1.000</td>
<td>0</td>
</tr>
<tr>
<td>Hub TE</td>
<td>1.832</td>
<td>1.190</td>
<td>1.213</td>
<td>-20</td>
</tr>
<tr>
<td>Case TE</td>
<td>1.829</td>
<td>1.330</td>
<td>1.213</td>
<td>-16</td>
</tr>
</tbody>
</table>
The computational domain needed to be partitioned so that all the processors could each iterate on a single block of the domain simultaneously. The flow solver uses Message Passing Interface (MPI) libraries for interprocessor communications. The architecture paradigm of the flow solver is distributed memory computation, so each partition, or block, has access to its own dedicated memory. A custom code, ‘jumbalaya’, was used to partition the domain into blocks with high parallel efficiency in mind in order to reduce computational time.

Partitioning was performed on a blade passage basis by selecting the number of partitions in the axial flow direction and in the radial direction. Due to the constraints of the flow solver the number of partitions in the radial direction needed to be constant for all blade rows, but no such constraint existed for the axial partitions. The procedure for partitioning the domain for high parallel efficiency included determining how partitioning each blade passage would affect the overall node counts in individual blocks. A sample of the partitioning details for mid spacing at $0.0^\circ$ stagger is given in Table 3.3. The goal of partitioning was to generate blocks with approximately the same number of nodes per block with the given constraints, as this would theoretically result in better parallel efficiency for the flow solver. In Table 3.3 it can be seen that the average number of nodes per block is nearly constant for each blade row.

The partitioning details for all the current simulations are given in Appendix B. Details on using the custom code “jumbalaya.x” are given in Appendix ?? in a tutorial format written for future students performing this type of research.

<table>
<thead>
<tr>
<th></th>
<th>Axial</th>
<th>Radial</th>
<th>No. Blades</th>
<th>No. Blocks</th>
<th>Avg. Nodes/Block</th>
</tr>
</thead>
<tbody>
<tr>
<td>Swirler</td>
<td>5</td>
<td>3</td>
<td>8</td>
<td>120</td>
<td>292,000</td>
</tr>
<tr>
<td>Deswirler</td>
<td>8</td>
<td>3</td>
<td>8</td>
<td>192</td>
<td>300,000</td>
</tr>
<tr>
<td>Rotor</td>
<td>10</td>
<td>3</td>
<td>7</td>
<td>210</td>
<td>293,000</td>
</tr>
<tr>
<td><strong>Total</strong></td>
<td></td>
<td></td>
<td></td>
<td><strong>522</strong></td>
<td></td>
</tr>
</tbody>
</table>
3.4 Numerical Solutions

The parallel flow solver, TURBO [43], was used to solve the unsteady Reynolds-Averaged Navier-Stokes equations using the NASA/CMOTT $k$-$\epsilon$ turbulence model specifically developed for turbomachinery flows by Zhu and Shih [44]. TURBO integrates to the wall for $y^+ < 10.5$, and uses wall functions for higher values. The turbulence model includes a near-wall damping term, which should justify the use of the $k$-$\epsilon$ model in this region. The solution algorithm is an iterative, implicit, time-accurate scheme with a finite-volume spatial discretization. The code utilizes two levels of subiteration schemes at each time step. An approximate Newton scheme is used to solve the nonlinear, discretized forms of the governing equations for the outer subiteration scheme, while a symmetric Gauss-Seidel relaxation scheme is used for the inner subiterations. Further details of the solution scheme are given in references [45] and [46], and the TURBO input files used for these simulations are given in Appendix D. The input files have details on the numerical schemes used for the simulations, as well as boundary conditions, reference values, and output parameters.

The TURBO code has been validated on numerous unsteady applications ranging from multistage compressors [47], stall inception [48, 49], distortion transfer [50, 51], blade-row interactions [13, 20, 38], and turbine aerodynamics [52, 53]. The code was developed specifically for turbomachinery applications, and is especially suited to study blade-row interactions.

The boundary conditions used in these simulations are consistent with those used by List [19]. The inlet boundary condition was an isentropic inlet with the same temperature, pressure, and velocity profiles measured at the inlet of the experimental BRI rig, with an inlet turbulence intensity of 2%. A no-slip condition was imposed at the hub, case, and on each blade surface. A sliding interface was imposed at the interface between the deswirler and rotor, allowing for direct simulation of the rotor bow shock/stator interaction. More details on the sliding interface used for this flow solver can be found in references [12, 13, 43]. The exit boundary condition was an imposed physical exit mass flow rate, corrected by density. It is known that using an imposed exit mass flow leads to slower convergence than using a pressure-based exit boundary condition, but justification for this exit boundary condition is given in the next section.
3.5 Simulation Convergence

An exit mass flow rate corresponding to peak experimental efficiency for each configuration of the BRI rig is imposed at the rotor exit for each simulation. Convergence was determined by examining the mass flow rates into and out of each blade row. Usually when the inlet and exit mass flow rates for each blade row converge to the imposed exit mass flow rate, the simulation is considered converged, and time-averaged and time-accurate solutions are generated for post-processing. In these simulations the mass flow rates rarely converged completely, but a periodic, low-frequency oscillation in the mass flow rates was observed. When the oscillation reached a periodic state, then the simulation was considered converged.

Using an exit mass flow rate for the exit boundary condition results in slower convergence, however the flow solver seemed to have difficulty using other exit boundary conditions. Using an exit pressure boundary condition can speed up convergence, but some iteration is needed in order to achieve the desired exit mass flow. Theoretically, if the mass flow is too low, the back pressure can be lowered in order to increase the mass flow. Attempts were made to use a pressure-based exit boundary condition, but this resulted in simulation divergence.

Experimentally-measured mass flow rates were available for each configuration. Attempts were also made to use an exit corrected mass flow (instead of a physical mass flow) because the solution converged much more quickly, but the drawback of this exit boundary condition was that the mass flow did not converge to the imposed value. When attempts were made to change the value of the exit mass flow, the flow field did not respond to these changes. It is still unknown why this occurs, but it may be related to the implementation of the exit boundary conditions within the code itself.

The current simulations were thus performed by imposing a physical exit mass flow corrected by density at the exit. This resulted in many rotor revolutions (from 15-25) until convergence, but allowed for time-averaged and time-accurate results at the desired mass flow rates and flow conditions. Table 3.4 gives the mass flow rates that correspond to peak experimental efficiency at each configuration. These values were the target mass flow rates used for each simulation.

With large-scale CFD simulations accelerating convergence is very important. Most of these simulations converged in roughly 500,000 CPU hours. A few different strategies were
Table 3.4: Mass Flow Rates at Peak Experimental Efficiency in kg/s.

<table>
<thead>
<tr>
<th>Axial Spacing</th>
<th>Stagger Angle</th>
<th>-3.0°</th>
<th>0.0°</th>
<th>+1.5°</th>
</tr>
</thead>
<tbody>
<tr>
<td>Close</td>
<td>14.846</td>
<td>—</td>
<td>14.567</td>
<td></td>
</tr>
<tr>
<td>Mid</td>
<td>14.892</td>
<td>14.703</td>
<td>14.594</td>
<td></td>
</tr>
<tr>
<td>Far</td>
<td>14.841</td>
<td>14.644</td>
<td>—</td>
<td></td>
</tr>
</tbody>
</table>

employed in order to accelerate convergence. All methods resulted in convergence, but some methods resulted in nearly 800,000 CPU hours. The method that obtained the fastest convergence is explained here. A plot of a typical mass flow history is given in Figure 3.5, with each region of interest explained in the following text.

Convergence was composed of five parts: initial transients (IT), initial convergence (IC), low-accuracy convergence (LAC), increased-accuracy (IA), and decreased time step (DTS), which are illustrated in Figure 3.5. In the initial transients (IT), time-marching iterations on the initialized flow field begin, and large changes to the flow field are encountered. In the initial convergence (IC) and low-accuracy convergence (LAC) the flow field is quickly brought closer to convergence with a low order solver. In increased-accuracy (IA) and decreased time step (DTS) the flow solver order is increased, and iterations continue. The following information describes each region in Figure 3.5:

- Initial Transients (IT): The simulation started with the initialized flow field, using 640 time steps per rotor blade passing, with the spatial and temporal accuracy at 2nd and 1st order accuracy for approximately a half revolution. This allowed for the simulation to step through the initial transients efficiently. This corresponds approximately to iterations 1 to 9,000 in Figure 3.5.

- Initial Convergence (IC): At this point the number of time steps per rotor blade passing was decreased to 320 for another half rotor revolution with the same spatial and temporal accuracy. This is shown in Figure 3.5 for iterations from 9,000 to 14,000. The simulation has transitioned from the initialized, interpolated, steady flow field to the unsteady flow field.
• Low-Accuracy Convergence (LAC): The number of time steps per blade passing was further decreased to 160 until the low frequency oscillation in the mass flow rates decayed sufficiently. This took anywhere from 10-15 full rotor revolutions, depending on the simulation. This corresponds approximately to iterations 14,000 to 38,000 in Figure 3.5.

• Increased-Accuracy (IA): When the amplitude of the oscillation in the mass flow rates had decayed sufficiently (usually <0.2 kg/s) the spatial and temporal accuracy were increased to 3rd and 2nd order respectively and run for approximately a half to a full rotor revolution. This caused the amplitude of the low-frequency oscillation of the mass flow rates to increase significantly, as seen around 40,000 iterations in Figure 3.5. Although the amplitude of the low-frequency oscillation remained high, this region lasted until approximately 60,000 iterations.

Figure 3.5: Mass Flow Rate History for Far Spacing at Decreased Loading. Note: IT=Initial Transients, IC=Initial Convergence, LAC=Low-Accuracy Convergence, IA=Increased-Accuracy, DTS=Decreased Time Step.
• Decreased Time Step (DTS): The number of time steps per rotor blade passing was then increased back to 320 to more accurately resolve the flow features of interest, and the simulation was ran for approximately a full rotor revolution. At that point the simulation was usually declared converged due to a periodic mass flow behavior explained below, and time-averaged and unsteady output was generated. The period of the low-frequency oscillation increases around 60,000 iterations in Figure 3.5, marking the decreased time step.

A low-frequency oscillation in the mass flow rates was observed for every simulation. It is thought that this is partially a byproduct of the exit boundary condition combined with the small axial region between the rotor exit and the exit plane. A low-frequency oscillation in the mass flow rate is often encountered in simulations due to the reflection of the exit boundary condition across the computational domain. One way to dampen the oscillation is to extend the computational domain farther downstream. This oscillation is also seen in simulations with a high amount of swirl at the exit. This problem could not be remedied for these simulations, since the rotor imposes a significant amount of swirl to the flow. In the current simulations the low-frequency oscillation never decayed completely, so the simulation was considered converged when a periodic state was reached. This usually occurred within a few rotor revolutions after the time step was decreased (region DTS from Figure 3.5).

Since these are time-accurate simulations, there should be some oscillation in the mass flow rates, as this is what occurs in turbomachines. It is not known how much oscillation is present in the BRI test rig. The low-frequency oscillation of the mass flow rates in these simulations is not a cause for concern, but the time-averaged and time-accurate solution generation was performed carefully in order to reduce the effects of the oscillation.

Due to the low-frequency oscillation in the mass flow rates the time-averaged results were obtained over a half period of the oscillation centered on the target mass flow rate, which was approximately a full rotor revolution. This is shown in Figure 3.6, which shows the mass flow history for the time-averaged solution output over a half period of the low-frequency oscillation for mid spacing at nominal loading. This was done in order to minimize the effects of the changing mass flow rate during the time-averaging process. It should be noted that for all of the simulations but far spacing at decreased loading, the mass flow rates during the time-averaged
solution generation were within 0.3 kg/s, or 2%, of the target mass flow rate, with some less than 0.15 kg/s, or 1%. For more details, see Appendix C.

Unsteady output was generated for a quarter annulus at every 20 time steps, resulting in 16 time steps per blade passing. Great care was taken during this step to generate unsteady data when the mass flow rates were closest to the target mass flow rate. This is shown in Figure 3.7, which shows the mass flow rate history for the unsteady solution output for mid spacing at nominal loading. The mass flow rates are all within 0.2 kg/s (less than 1.5%) of the target mass flow for a quarter of a revolution, allowing for time-accurate results at the desired operating condition. Mass flow rate histories for all the simulations in this analysis are given in Appendix C.
Figure 3.7: Mass Flow Rate History for the Unsteady Solution Generation for Mid Spacing at Nominal Loading.
CHAPTER 4. RESULTS

Results from the numerical simulations are given in this chapter. Time-averaged results showing the simulation efficiency and the deswirler pressure distribution are given. Time-accurate results of the pressure distribution are presented to give insights into the effects of the rotor bow shock on the deswirler blade. The suction surface boundary layer behavior is explained by examining the velocity, vorticity, and pressure gradient throughout the entire deswirler passage in order to explain the formation of a low-velocity region that forms at the deswirler trailing edge. The emphasis of this chapter is on vortex shedding, and contour plots of representative unsteady data are given to illustrate vortex shedding phenomenon. Vortex strength is detailed, with the development of a model to help predict shock-induced vortex strength due to stator loading and shock strength.

4.1 Time-Averaged Efficiency

The time-averaged efficiencies help validate the simulations. It is common for CFD simulations to overpredict efficiencies compared to experimental results, but if trends are matched, then useful conclusions can be made from the data. These simulations exhibit an overprediction of the efficiencies, but match the experimental trends well.

Time-averaged solutions were obtained over a full rotor revolution in order to minimize effects of a low-frequency oscillation in the mass flow rates caused by the exit boundary condition reflecting across the computational domain (see Section 3.5). The time-averaged efficiencies for each simulation were calculated from the domain inlet and exit total temperatures and pressures, as shown by Equation 4.1. The calculated efficiencies were consistently 5% higher than the corresponding experimental efficiencies. List [19] found the same issue with his simulations, and the error was attributed to the lack of accounting for the losses incurred in the downstream stator. Another source of error could be the lack of the turbulence model to ac-
curately resolve shear flows in the boundary layers, which is a common difficulty for the $k$-$\epsilon$ turbulence model [54]. The calculated efficiencies at mid spacing for all three loadings are given in Figure 4.1 with the corresponding experimentally measured efficiencies at peak performance. Although the magnitudes of calculated efficiency are off by a factor of about 0.95, the same trends in the experimental efficiency are observed in the current simulations at mid spacing, instilling confidence in the simulations. It is remarkable how closely the experimental trends in efficiency are reproduced in the CFD results. Both experimental and computational results show decreased loading to have the highest efficiency, with nominal loading lower, and increased loading the lowest. The mass flow rates corresponding to peak efficiency are also predicted very well. This was partly due to the exit boundary condition forcing the exit mass flow rate to be the target mass flow rate, but the inlet mass flow rate was also matched because of how the low-frequency oscillation was managed (see Section 3.5 and Appendix C). It is expected that if more simulations were performed at other mass flow rates the curves in Figure 4.1 would be reproduced with the same trends. It should be noted that the simulation efficiencies in Figure 4.1 are scaled by a constant factor of 0.95 in order to show all data points in the same range, and show the comparison of the experimental and CFD efficiency trends.

$$\eta = \left( \frac{p_{te}}{p_{ti}} \right)^{\frac{\gamma - 1}{\gamma}} - 1$$

(4.1)

The time-average pressure ratios were also compared for the CFD and experimental results, and these are plotted against mass flow rate for each simulation in Figure 4.2, where solid symbols are the experimental values and open symbols are the CFD values. Similar to the time-averaged efficiency the CFD results are higher than the experimental results. This is partly due to the lack of the downstream stator in the simulations, as well as the turbulence model not accounting for all of the sources of loss in the rig. The trends observed in the CFD data do not match the experimental data as well as the time-averaged efficiency does. When comparing the effects of stator loading for a given axial spacing the trends match sufficiently. At close spacing, as loading increases both the CFD and experimental pressure ratios both decrease slightly. At mid spacing, as loading increases the experimental pressure ratio decreases, while the CFD pressure ratio increases, then decreases. At far spacing both the experimental and CFD pressure
ratios decrease with increased stator loading. It can be seen in Figure ?? that at peak efficiency for each simulation, both the CFD and experimental pressure ratios changed very little. It appears that pressure ratio for these configurations does not have a strong dependency on stator loading or axial spacing.

4.2 Stator Loading

Since stator loading is one of the main features of interest for these simulations this section describes the stator loading for both the time-averaged and unsteady results. The time-averaged stator loading is commonly used for design purposes, but the unsteady stator loading reveals further insights into the static pressure behavior. All of these results are given at midspan.
4.2.1 Time-Averaged

The time-averaged pressure distribution at midspan was calculated for the simulations at mid spacing in order to quantify loading change, and is shown in Figure 4.3. $C_p$ was calculated from Equation 4.2, where $\overline{p_1}$ is the deswirler inlet average static pressure and $\overline{p_{t1}}$ is the inlet average total pressure at midspan. These values were found for each simulation and are given in Tables 4.1 and 4.2. For the most part these values are very similar for each simulation. The total pressure values are all essentially the same, but the static pressure values decrease slightly with increased stator loading. From decreased to nominal loading the static pressure values decrease by 3.5% and from nominal to increased loading the static pressure values decrease by 1.7%.

$$C_p = \frac{p - \overline{p_1}}{\overline{p_{t1}} - \overline{p_1}} \quad (4.2)$$

The pressure surface distribution is similar for all three loadings at mid spacing. On the suction surface, the nominal (0.0°) and increased loading (+1.5° stagger) profiles are sim-
Table 4.1: Inlet Average Static Pressure, $p_1$, Values in kPa.

<table>
<thead>
<tr>
<th>Axial Spacing</th>
<th>Stagger Angle</th>
<th>(-3.0^\circ)</th>
<th>(0.0^\circ)</th>
<th>(+1.5^\circ)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Close</td>
<td>73.6</td>
<td>71.3</td>
<td>69.9</td>
<td></td>
</tr>
<tr>
<td>Mid</td>
<td>72.2</td>
<td>70.3</td>
<td>69.3</td>
<td></td>
</tr>
<tr>
<td>Far</td>
<td>74.3</td>
<td>70.8</td>
<td>—</td>
<td></td>
</tr>
</tbody>
</table>

Table 4.2: Inlet Average Total Pressure, $p_{t1}$, Values in kPa.

<table>
<thead>
<tr>
<th>Axial Spacing</th>
<th>Stagger Angle</th>
<th>(-3.0^\circ)</th>
<th>(0.0^\circ)</th>
<th>(+1.5^\circ)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Close</td>
<td>100.7</td>
<td>100.7</td>
<td>100.6</td>
<td></td>
</tr>
<tr>
<td>Mid</td>
<td>100.2</td>
<td>100.9</td>
<td>100.6</td>
<td></td>
</tr>
<tr>
<td>Far</td>
<td>100.7</td>
<td>100.3</td>
<td>—</td>
<td></td>
</tr>
</tbody>
</table>

ilar. The major difference between those and decreased loading (-3.0° stagger) is the greater flow deceleration between 10% and 40% chord which would generate a thicker boundary layer farther upstream. At decreased loading the stator is substantially less loaded on the first 20% chord (where $C_x$ is defined as the chord length of the deswirler). Integration of $C_p$ over $x/C_x$ yields stator loading values of 0.990, 1.102, and 1.123 for decreased, nominal, and increased loading respectively. This corresponds to an increase from decreased to nominal loading by 11.3%, and from nominal to increased loading by 2.1%.

4.2.2 Unsteady

Hereafter the time-accurate results will be presented. The time-average solutions have been shown to match the experimental trends, and the unsteady solutions will be shown in Chapter 5 to match the experimental PIV trends as well. The circulation of the shed vortices in the CFD simulations will be shown to closely match the PIV circulation results.

Figure 4.4 shows the unsteady blade loading of the deswirler at midspan for mid spacing at nominal loading. Solid lines are time-averaged values, dashed lines are the time-averaged values plus the standard deviation of the unsteady pressure coefficient, and dotted lines are
the time-averaged values minus the standard deviation of the unsteady pressure coefficient. Analysis of instantaneous blade loading for one quarter rotor revolution showed the greatest loading variation occurred at the pressure surface of the trailing edge, a consequence of the rotor bow shock impacting the stator trailing edge. This is shown in Figure 4.4 by the vertical black lines between 0.60 and 1.00 $x/C_x$.

A sample of an instantaneous plot of the pressure distribution is given in Figure 4.5 for mid spacing at nominal loading. The rotor bow shock has just impacted the trailing edge, and a large pressure jump at 0.95 chord results. The strength of this shock decays as it propagates upstream, and previous rotor bow shocks can be seen at 0.45 and 0.20 chord on the pressure
The suction surface is characterized by multiple pressure waves that are reflected through the deswirler passage. This phenomenon will be described in further detail in Section 4.3.3.

### 4.3 Suction Surface Boundary Layer Behavior

Initial analysis of the current simulations focused on the vortex shedding at the stator trailing edge. The time-accurate data showed large variations in vortex shedding patterns between the 8 stator blade passages. The variation between blade passages was a consequence of the swirler wake passing downstream and bow shock propagation upstream through the sta-
Figure 4.5: Instantaneous Pressure Coefficient Over the Stator Chord for Mid Spacing at Nominal Loading.

The wake velocity defect on the suction surface and downstream of the trailing edge was observed to have a significant impact on shed vortex strength. Larger and stronger vortices formed as a result of a low-velocity region that formed at the suction side of the deswirler trailing edge. This low velocity region allowed vortex formation to take place over a longer time, resulting in more pressure side, negative vorticity fluid to be ingested in the vortex before roll-off. This effect is amplified at increased loading, which causes these larger vortices to become even stronger at increased stator loadings. Hereafter, “normal” vortices will refer to shock-induced vortices that appear to form normally at the deswirler trailing edge due to the shock impingement event, and “large” vortices will refer to the shock-induced vortices that form in a low-velocity region at the deswirler trailing edge due to the shock impingement event.
It was hypothesized that this low-velocity region began as separated flow that propagated downstream from upstream of midchord. The existence of a large, separated flow region at the deswirler trailing edge was first observed with PIV measurements of the BRI rig [17]. These numerical simulations as well as those of List [20] have shown such periodic behavior. Additional evidence of similar behavior on a different compressor was reported by Sanders and Fleeter [15]. The low velocity region on the suction surface was attributed to a shock-induced thickening of the boundary layer, or formation of a separation bubble. As the shock propagates upstream and reflects through the deswirler passage it causes flow non-uniformities on the blade that can cause suction surface boundary layer separation. Results are now presented that show the details of the formation of a large low velocity region on the stator suction surface due to the the deswirler suction surface boundary layer behavior. The suction surface boundary layers for mid spacing at nominal and decreased loadings are presented here.

4.3.1 Boundary Layer at Nominal Loading

Figure 4.6 presents a sequence of images showing a typical low-velocity region forming upstream on the deswirler suction surface then propagating and growing downstream to the trailing edge at nominal loading. Velocity contours help to visualize the rotor bow shock as it propagates upstream along the deswirler. Vorticity contour lines help to visualize both the boundary layers and the vortex formation. The time where the rotor bow shock first interacts with the deswirler trailing edge is defined as $t/T = 0$, where $t$ is time, and $T$ is the time between shock passing events. A value of $t/T = 1$ is one rotor bow shock passing event after $t/T = 0$. A low-velocity region originates between 0.10 and 0.20 chord for nominal loading, as a small increase in the laminar boundary layer thickness, as seen in Figure 4.6a. In Figure 4.6b a low-velocity bubble at 0.30 chord is observed to grow—possibly a separation bubble that transitions the boundary layer from laminar to turbulent. Figure 4.6c shows the bubble at 0.60 chord has grown substantially in size. A second rotor bow shock has just interacted with the deswirler trailing edge. In Figure 4.6d the low velocity region arrives at the trailing edge of the deswirler, and a third rotor bow shock is interacting with the trailing edge, while in Figure 4.6e a “large” vortex begins to form in this low-velocity region. The low-velocity regions at nominal loading remained at the trailing edge on average for 25% longer than those at decreased loading. With
the additional separation, this region moves slower and it takes more freestream fluid to “push” it downstream.

Figure 4.6: Contours of Axial Velocity with Lines of Vorticity at Midspan for Nominal Loading, Showing the Formation of the Low-Velocity Region.
4.3.2 Boundary Layer at Decreased Loading

The behavior of the stator suction surface at decreased loading is very different than that at nominal loading, though some large vortices are still formed for this configuration. At decreased loading only 24% of the measured vortices were considered “large” while at nominal loading 58% were “large”. A negligible amount of suction side separation was observed for decreased loading on the first 20-30% of the deswirler chord. Most of the suction surface boundary layer thickness increase was observed around 0.55 chord. This is because at decreased loading the boundary layer on the leading edge of the deswirler is much thinner than at nominal loading, causing it to separate much farther downstream. The boundary layer at nominal loading is thicker than decreased loading near 0.20-0.30 chord, so the possible separation is observed much farther upstream.

Figure 4.7 shows the formation of the low velocity region at the deswirler suction side trailing edge for decreased loading. At decreased loading the low-velocity regions appear as an increase in the suction side boundary layer thickness around mid chord. In Figure 4.7a a pressure wave near 0.55 to 0.65 chord appears to cause the suction side boundary layer thickness to grow, and form a low-velocity bubble. This propagates downstream and grows in pitch with each pressure wave. In Figure 4.7b the bubble at 0.65 chord is seen to grow further. In Figure 4.7c the bubble grows even more in the pitchwise direction near 0.80 chord, and starts to form a low-velocity region at the trailing edge. In Figure 4.7d the bubble near 0.90 chord strengthens as a rotor bow shock passes and causes the flow to decelerate. At $t/T = 1.975$ in Figure 4.7e the low-velocity region begins to roll-off the trailing edge, while a “large” vortex begins to form in this low-velocity region.

4.3.3 Effects of Pressure Gradient

Figure 4.8 shows the normalized two-dimensional pressure gradient magnitude (described by Equation 4.3) of the same images shown in Figure 4.6, and shows a very complex pressure field in the stator passage. Equation 4.3 describes the pressure gradient magnitude in the axial and tangential directions at a constant radius, where $p_{ref} = 101.3$ kPa and $C_x = 4.852$ cm is the axial chord length of the deswirler. The results presented here show the two-
Figure 4.7: Contours of Axial Velocity with Lines of Vorticity at Midspan for Decreased Loading.

dimensional pressure gradient at midspan.

$$NPG \equiv \left( \frac{\partial (p_s / p_{ref})}{\partial (x/C_x)} \right)^2 + \left( \frac{\partial (p_s / p_{ref})}{\partial (S/C_x)} \right)^2 \right)^{1/2} \quad (4.3)$$

The pressure gradient contours clearly show the pressure waves, and the vorticity lines effectively show the boundary layer thickness and vortex formation. The deswirler passage contains remnants of the rotor bow shock that propagates upstream on the deswirler pressure and
suction surface as well as reflected pressure waves from multiple previous bow shocks. In Figure 4.8a the rotor bow shock impacts the stator trailing edge, a pressure wave from the previous rotor bow shock can be seen at 0.40 chord on the pressure surface, and numerous pressure waves (see 0.10, 0.40, and 0.80 chord) on the stator suction surface are also present. In Figure 4.8b the pressure wave at 0.30 chord causes the suction side boundary layer to thicken. In Figure 4.8c the rotor bow shock reflects off the pressure surface of the deswirler at 0.90 chord, causing a reflected shock to propagate upstream and toward the deswirler suction surface that
is located above. Similar reflections can also be seen in the other images. A similarly reflected wave interacts with the suction surface in Figure 4.8a at 0.80 chord, Figure 4.8b at 0.65 chord and in Figure 4.8c at 0.80 chord. These reflected waves also propagate upstream on the stator suction surface, eventually reaching the leading edge. As the pressure waves approach the deswirler curvature at 0.30 chord the waves begin to accelerate and coalesce near the leading edge (between 0.0 and 0.20 chord in Figures 4.8d-e). These pressure waves appear to be strengthened by the acceleration on the fore section of the front-loaded deswirler.

The boundary layer and separation bubble characteristics observed in Figure 4.6 can be related to the pressure waves that propagate upstream from the rotor bow shock. It was observed that the deswirler pressure surface pressure wave has an effect on the suction surface boundary layer thickness at the leading edge. As the pressure wave approaches the leading edge it refracts around the sharp corner of the leading edge and increases the boundary layer thickness at the leading edge, which effectively increases the radius of the leading edge. This upsets the suction surface pressure gradient at the leading edge, and thus increases the suction surface boundary layer thickness between 0.0 and 0.15 chord.

Figure 4.9 displays a sequence of images of pressure gradient magnitude contours and vorticity lines that show the pressure side pressure wave affecting the suction surface boundary layer thickness near the leading edge. Figure 4.9a shows a pressure surface wave just before it reaches the leading edge. The suction surface boundary layer is small and unaffected by the pressure wave. In Figure 4.9b the pressure wave is at the stator leading edge and in Figure 4.9c the pressure wave has just passed out of frame. In Figure 4.9c the boundary layer thickness begins to grow at the leading edge. Figure 4.9d shows the suction surface boundary layer thickness growing and possibly beginning to separate from the blade, as shown by the vorticity lines close to the blade surface near the leading edge.

Figure 4.10 shows the suction side pressure waves combining near the leading edge to create a strong, localized, adverse pressure gradient, causing the leading edge boundary layer to separate. Figure 4.10a shows suction side pressure waves combining at about 0.15 to 0.20 chord. In Figure 4.10b these waves combine to form a stronger pressure jump, and the suction side boundary layer thickness begins to increase. In Figure 4.10c the suction side boundary layer thickness continues to grow, and the pressure wave remains at 0.20 chord on the deswirler.
suction surface. This causes the boundary layer to separate in Figure 4.10d, and in Figure 4.10e this separation bubble propagates downstream. It should be noted that this occurs independently of the pressure side pressure wave.

At times the effects of the pressure side and suction side pressure waves combine, and the effect is a large separation that results in a large low-velocity region downstream. This is shown in Figure 4.11. In Figures 4.11a-b the pressure side pressure wave changes the leading edge flow conditions, and causes the suction side boundary layer thickness to increase. The suction side pressure wave at 0.20 chord causes it to grow even more. Figure 4.11c shows the suction side pressure wave strengthening, similar to Figure 4.10b, and the boundary layer beginning to separate in the region of the strong, localized, adverse pressure gradient. As the separated boundary layer passes through the strengthened suction side pressure wave it separates...
even more, as shown in Figure 4.11d. Figure 4.11e shows the separation bubble propagating farther downstream and growing even more as it passes through another pressure wave around 0.30 chord. This results in a strong low-velocity region at the deswirler suction side trailing edge at $t/T = 2.75$ (not shown here).

![Images](a) $t/T = 0.00$  (b) $t/T = 0.125$  (c) $t/T = 0.25$  (d) $t/T = 0.375$  (e) $t/T = 0.50$

Figure 4.11: Contours of Normalized Pressure Gradient Magnitude (Same Levels as in Figure 4.8) with Lines of Vorticity at Midspan for Nominal Loading for the first 30% deswirler chord Showing the Combined Effects of a Passing Pressure Wave on the Deswirler Pressure Side and Another on the Suction Side.

The apparent thickening of the boundary layer observed in Figures 4.6a-b upstream of 0.20 chord is caused by the combination of a pressure surface pressure wave reaching the stator leading edge and a pressure gradient on the suction surface by the reflecting pressure waves around 0.10 to 0.20 chord, a phenomenon which is shown in Figure 4.8a. The separation bubble at $t/T = 0.375$ of Figure 4.6b coincides with the suction surface pressure wave at 0.30 chord. The growth of the bubble observed at $t/T = 1.1875$ in Figure 4.6c also occurs where the reflected rotor bow shock interacts with the suction surface of the deswirler at 0.75 chord.

To summarize this phenomenon, a reflected wave from an adjacent blade disturbs the suction surface boundary layer more than a single pressure wave moving upstream, since it propagates directly at the suction surface boundary layer. As this pressure wave reaches the blade above, it reflects off the blade above around 0.20 chord, which can cause an increase to the suction surface boundary layer thickness. This perturbation can cause a low-velocity bubble to form on the fore section of the deswirler, which propagates downstream and eventually grows into a large low-velocity region at the deswirler trailing edge. The pressure surface pressure wave can also cause the suction side boundary layer to increase as it propagates upstream and refracts around the leading edge of the stator.
4.4 Vortex Shedding

Time-accurate data at midspan was generated for one quarter rotor revolution. This allowed for analysis of approximately 50 rotor blade passings for each simulation. As in other experimental and CFD blade-row interaction studies it was observed that vortex shedding was phase-locked with the passing of the rotor bow shock [10–13, 16, 20]. It was verified that the mechanism responsible for causing the vortex formation was a change in unsteady loading on the deswirler (see Figure 4.5) caused by the passing rotor bow shock, showing that vortex generation was phase-locked with the rotor passing. Each time the rotor bow shock impacted the deswirler trailing edge a shock-induced vortex formed at the trailing edge and convected downstream.

Examination of the time-accurate data showed large variations in vortex shedding patterns between the 8 blade passages. Such variation could not be observed in the PIV experiment since images were only obtained at one stator trailing edge. In the experimental PIV analysis of Reynolds [16], a consistent pattern of shed vortices, with three distinct vortices shed per blade passing was observed. In the current simulations it was also observed that a strong vortex was shed due to the passing rotor bow shock. The variation between blade passages has been shown to be a consequence of the unsteady flow field and shock propagation upstream through the stator passage. This flow non-uniformity will be shown to be a major contributor to the size and strength of shock-induced vortices.

4.4.1 “Normal” Vortex Formation

Figures 4.12, 4.13, and 4.14 illustrate the formation of typical shock-induced vortices at mid spacing at decreased, nominal, and increased loading respectively. Velocity contours help to visualize the rotor bow shock as it propagates upstream through the deswirler passage, and vorticity contour lines help to visualize both the boundary layers and the vortex formation. Again, \( t/T = 0 \) is defined as the time when the rotor bow shock first interacts with the deswirler trailing edge.

The vortex formation occurs in the following manner as illustrated in Figures 4.12-4.14. In Figures 4.12b, 4.13b, and 4.14b the passing bow shock causes the stator wake to deflect
slightly in the direction of the rotor rotation (down). The pressure side boundary layer contains a very strong negative vorticity component, as the steepest velocity gradients are encountered there. The fluid from the pressure side boundary layer flows downstream, forming a small bub-
Figure 4.13: Axial Velocity Contours with Vorticity Lines Showing “Normal” Vortex Formation on Deswirler TE at Midspan for Mid Spacing at Nominal Loading.

A bubble of strong negative vorticity on the deswirler trailing edge. The bubble grows in pitch and axial extent until the suction side fluid finally pinches off the vortex, and causes it to roll-off the trailing edge and propagate downstream, as shown in Figures 4.12c, 4.13c, and 4.14c. In Fig-
Figure 4.14: Axial Velocity Contours with Vorticity Lines Showing “Normal” Vortex Formation on Deswirler TE at Midspan for Mid Spacing at Increased Loading.

ures 4.12e, 4.13e, and 4.14e the vortex is well-defined near 20% deswirler chord downstream, where vortices were compared for all three loadings.
These vortices will be referred to hereafter as the “normal” vortices. Figure 4.12 shows a sequence of images illustrating the typical formation of a “normal” vortex on the deswirler trailing edge at midspan at decreased loading, Figure 4.13 shows the same for nominal loading, and Figure 4.14 shows the same for increased loading. All of these image sequences show a comparatively thin suction surface boundary layer thickness, which was typical for “normal” vortex formation. This allows the higher momentum freestream fluid to help push the shock-induced vortex off the trailing edge before it grows too large, as shown by the small pitchwise extent of the low velocity region at the deswirler trailing edge at $t/T = 0.25$ in Figure 4.12b and $t/T = 0.3125$ in Figures 4.13b and 4.14b.

It is surprising that the strength of these “normal” vortices is nearly constant for mid spacing for the three different stator loadings. These figures show that the behavior of the “normal” vortices is very similar for all three configurations, despite having different stator loadings. It should be noted that these “normal” vortices are categorized in a qualitative manner, and are characterized by a small suction surface boundary layer. The formation of other “large” vortices is described in the next section, and the strength of these vortices does change with stator loading. These results will be compared with the results in the next section.

Vorticity contours were also analyzed for the same sequence of images to help interpret the results shown in Figures 4.12-4.14, but are not included here for brevity. They are included in Appendix A.

4.4.2 “Large” Vortex Formation

The velocity defect on the suction surface and downstream of the trailing edge was observed to have a significant impact on shed vortex strength. Figures 4.15, 4.16, and 4.17 show a different mode of vortex formation for decreased, nominal, and increased loading at mid spacing respectively. Here, as the vortices began to form, they were sometimes strengthened by a severe velocity defect in the wake region that originated from the suction surface boundary layer between 0 and 50% chord. This is evident from the low velocity region at the suction side trailing edge (last 20% chord) in Figures 4.15a-b, Figures 4.16a-c, and in Figures 4.17a-b. When vortices were seen to form in these regions of severe velocity defect at the trailing edge, the measured strength was significantly higher than that of the “normal” vortices. As was mentioned
previously, these vortices are referred to as “large” vortices. In Figures 4.15d, 4.16d, and 4.17d a much larger vortex begins to roll-off the trailing edge, and in Figures 4.15e, 4.16e, and 4.17e the vortex is shown at 0.20 chord downstream where the circulation was measured.

The source or cause of this severe velocity defect region was shown in Section 4.3 to be a periodic thickening of the boundary layer or separation bubble that forms far upstream on the deswirler. At decreased loading this boundary layer thickening occurred near 45% chord, and at nominal loading it occurred near 15-20% chord. This is due to the increased incidence on the deswirler. At increased incidence the boundary layer is much thicker than at design incidence at the same location upstream. This causes the boundary layer to be thicker for nominal loading at 15-20% chord than decreased, which thickens farther downstream, at 45% chord. This suction side boundary layer thickening and separation was observed during borescope PIV measurements of the BRI rig as well [17], which were able to capture both suction and pressure surfaces of the deswirler trailing edge. These numerical simulations as well as those of List [20] have shown such periodic behavior. This occurs when the suction surface boundary layer thickens, and sometimes separates when the rotor bow shock propagates upstream. Additional evidence of similar behavior was reported by Sanders and Fleeter [15].

It should be noted that Figures 4.12 and 4.15 contrast the formation of “normal” and “large” vortices at decreased loading, Figures 4.13 and 4.16 contrast “normal” and “large” vortices at nominal loading, and Figures 4.14 and 4.17 contrast the formation of “normal” and “large” vortices at increased loading. The low-velocity region at the suction surface and downstream of the trailing edge is much larger and more severe in Figures 4.15-4.17, which causes stronger vortices to be shed from the deswirler pressure side. This low velocity region has a large pitchwise extent, and the low velocity fluid in the suction surface trailing edge region does not push the vortex downstream quickly, but rather allows the vortex formation to take place over a longer time interval (as noted by the time stamp in each image). “Normal” vortices were near 20% chord downstream at \( t/T = 0.6875 \) and \( t/T = 0.625 \) in Figures 4.12 and 4.13 respectively, while “large” vortices were not located near 20% chord until the next rotor bow shock was close to interacting with the deswirler trailing edge at \( t/T = 0.9375 \) in Figures 4.15 and 4.16. This allowed more pressure side boundary layer fluid to be ingested into the “large” vortices before
roll-off. This caused these vortices to be significantly larger and stronger than the “normal” vortices which form without the same severe velocity defect.
Figure 4.16: Axial Velocity Contours with Vorticity Lines Showing “Large" Vortex Formation on Desirer TE at Midspan at Nominal Loading.
Figure 4.17: Axial Velocity Contours with Vorticity Lines Showing “Large” Vortex Formation on Deswrler TE at Midspan at Increased Loading.
CHAPTER 5. CIRCULATION ANALYSIS AND MODEL DEVELOPMENT

This chapter presents the circulation results for each of the simulations analyzed. The CFD results compare well with the experimental data. Dimensional analysis was used in order to develop a model to predict shock-induced vortex circulation due to a change in blade loading and shock strength. A discussion of the relevant dimensional analysis is given, and the development of the model is described.

5.1 Circulation

Circulation was used as the measure of vortex strength for this analysis. The circulation was computed 20% chord downstream of the deswirler trailing edge because this is where the vortices were well-defined. The experimental data was also taken at 20% chord downstream. The circulation was calculated from the line integral of velocity for each well-formed vortex observed in the unsteady data in order to quantitatively compare vortices. Vortices were seldom circular, so a non-circular integration path was used at a zero-vorticity contour [55], as seen in Figure 5.1. The thick black line in Figure 5.1 shows the extracted data at an approximate zero-vorticity contour. The horizontal line that appears as a discontinuity in the center of the image is due to interpolation error onto the midspan plane for the unsteady results. This interpolation error is especially apparent for derivative quantities, such as vorticity. The effect of this interpolation error on the circulation results is negligible. The equation for circulation is given in Equation 5.1, where $\Gamma$ is in units of $m^2/s$. It should be noted that simulations for far spacing at increased loading and of close spacing at nominal loading were not analyzed for this research. Thus these results are not included in the tables that follow.

$$\Gamma = \iint \omega \cdot n \, dA = \oint \mathbf{u} \cdot d\mathbf{l} = \int u \, dx + v \, dy + w \, dz$$ (5.1)
Figure 5.1: Vorticity Lines and Contours with a Non-Circular Integration Path for the Circulation of a Large Vortex for Mid Spacing at Nominal Loading. This is the Same Vortex Seen in Figure 4.13e.

The average circulation values for both “normal” and “large” vortices at mid spacing are given in Tables 5.1 and 5.2 respectively, and are also plotted in Figure 5.2 for mid spacing. The “normal” vortex strength is relatively constant for a constant shock strength. These vortices all formed without a low-velocity region at the trailing edge. The results show that shock strength affects shed vortex strength. “Normal” vortices shed at far spacing are about 18% weaker than vortices shed at close and mid spacing. It is not possible to determine if “normal” vortices at mid spacing are weaker than those shed at close spacing, because of the lack of the one close data set, but the trend is seen from far to mid.

Table 5.1: Average “Normal” Vortex Strength for Shock-Induced Vortices with Units of $m^2/s$.

<table>
<thead>
<tr>
<th></th>
<th>−3.0°</th>
<th>0.0°</th>
<th>+1.5°</th>
</tr>
</thead>
<tbody>
<tr>
<td>Close</td>
<td>0.406</td>
<td>—</td>
<td>0.456</td>
</tr>
<tr>
<td>Mid</td>
<td>0.429</td>
<td>0.448</td>
<td>0.431</td>
</tr>
<tr>
<td>Far</td>
<td>0.345</td>
<td>0.365</td>
<td>—</td>
</tr>
</tbody>
</table>
This effect is much more important for the “large” vortices. These results are given in Table 5.2. It can be seen that “large” vortex strength increases with the rotor bow shock strength (decreased axial spacing) and with stator loading. This is consistent for every simulation. The circulation of the “large” vortices increases for far to mid spacing by 24% and from mid to close spacing by 7%.

At mid spacing the “large” vortices are 1.7, 1.9, and 2.1 times stronger than the “normal” vortices for decreased, nominal, and increased loading respectively. At close spacing the “large”
vortices are 1.9 and 2.2 times stronger than the “normal” vortices for decreased and increased loading respectively. At far spacing the “large” are greater than the “normal” by 1.7 and 1.9 times for decreased and nominal loading.

Since the focus of this research has been on quantifying the shock-induced vortex strength, Table 5.3 averages all of the measured shock-induced vortices for each simulation (both “normal” and “large”). It is important that all of the shock-induced vortices are combined into one data set in order to develop a model that help designers account for unsteady effects, such as shock-induced vortex shedding. The same trends observed in Table 5.2 are observed here: circulation increases with rotor bow shock strength (decreased axial spacing) and stator loading. Circulation is seen to increase from far to mid spacing by 21% on average, and from mid to close spacing by 20%. On average, circulation also increased by 19% from decreased to nominal loading and by 26% from nominal to increased loading.

Table 5.3: Average Circulation of All Shock-Induced Vortices (“Normal” and “Large”) in Units of m$^2$/s.

<table>
<thead>
<tr>
<th></th>
<th>−3.0°</th>
<th>0.0°</th>
<th>+1.5°</th>
</tr>
</thead>
<tbody>
<tr>
<td>Close</td>
<td>0.620</td>
<td>—</td>
<td>0.831</td>
</tr>
<tr>
<td>Mid</td>
<td>0.500</td>
<td>0.690</td>
<td>0.718</td>
</tr>
<tr>
<td>Far</td>
<td>0.434</td>
<td>0.542</td>
<td>—</td>
</tr>
</tbody>
</table>

Experimental PIV results of many of the same configurations for the BRI rig show similar trends. Reynolds [16] showed that shed vortex circulation increased with shock strength and stator loading. In that analysis the results were phase-averaged over 50 or more blade passings. The results showed two “strong” vortices, followed by a “weak” vortex for every rotor blade passing. The strong vortices were shock-induced, so they make a good comparison with the data presented in this analysis. These values are given in Table 5.4. There were no PIV data obtained at increased loading, or for close spacing at decreased loading. A comparison of mid and far spacing at decreased and nominal loading showed good agreement between the experimental results and the CFD results. The percent difference between the CFD and PIV results is given in...
Table 5.5. An average variation of 8.3% is seen between experimental and CFD results. The CFD results are slightly lower than the experimental results.

Table 5.4: Circulation for PIV Results of Shock-induced Vortices [16] in Units of $m^2/s$.

<table>
<thead>
<tr>
<th></th>
<th>$-3.0^\circ$</th>
<th>$0.0^\circ$</th>
<th>$+1.5^\circ$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Close</td>
<td>—</td>
<td>0.705</td>
<td>—</td>
</tr>
<tr>
<td>Mid</td>
<td>0.568</td>
<td>0.752</td>
<td>—</td>
</tr>
<tr>
<td>Far</td>
<td>0.462</td>
<td>0.580</td>
<td>—</td>
</tr>
</tbody>
</table>

Table 5.5: Percent Difference Between Experimental PIV Circulation Results (Table 5.4) and the Average of all the CFD Circulation Results (Table 5.3).

<table>
<thead>
<tr>
<th></th>
<th>$-3.0^\circ$</th>
<th>$0.0^\circ$</th>
<th>$+1.5^\circ$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mid</td>
<td>12%</td>
<td>8.3%</td>
<td>—</td>
</tr>
<tr>
<td>Far</td>
<td>6.1%</td>
<td>6.6%</td>
<td>—</td>
</tr>
</tbody>
</table>

Reynolds showed that when axial spacing changed from mid to far, circulation decreased by 19% and 23% for decreased and nominal loading respectively. These results show that when axial spacing changed from mid to far, the average shock-induced vortex circulation decreased by 13% and 21% respectively. The CFD results are slightly lower than the PIV results, but similar trends are observed. Reynolds also showed that when loading changed from nominal to decreased loading, circulation decreased by 25% and 20% for mid spacing and far spacing respectively. The current CFD results show that when loading was changed from nominal to decreased loading, the average circulation decreased by 28% and 20% for mid and far spacing respectively. These values are very close to those values found by Reynolds further validating the current time-accurate data sets.

The increase in circulation from “normal” to “large” vortices for a given configuration also increased with blade loading. On average circulation increased from “normal” to “large” vortices by a factor of 1.7 for decreased loading, 1.9 for nominal loading, and 2.1 for increased loading. This implies that there is more variation in vortex strength as loading increases. At
decreased loading the vortices were mostly “normal” with the occasional “large” vortex. 24% of the vortices measured at decreased loading for mid spacing were considered “large”. For mid spacing at nominal and increased loading there were more “large” vortices than “normal” vortices. At nominal loading the number of “large” vortices increased to 58% of the measured vortices, and at increased loading 54% of the measured vortices were considered “large”. As the deswirler loading increases, the interaction of the rotor bow shock contributes to the unsteady suction surface boundary layer separation, and causes it to occur more frequently, which results in a greater frequency of “large” vortices.

5.2 Dimensional Analysis

The ultimate goal of this research has been to develop a model to relate shed vortex strength to stator loading and axial spacing in order to allow compressor designers to account for these unsteady interactions. This section briefly describes the dimensional analysis and the process that was used to generate a model with the results of this analysis.

The results of this analysis have shown that blade loading is important to determining shed vortex circulation. Various parameters measure blade loading. The integration of static pressure across the stator most accurately measures blade loading, but diffusion factor was chosen as the parameter of interest since it is a commonly used parameter in compressor design. Diffusion factor was calculated from Equation 5.2, where $V_1$ is the velocity magnitude at midspan at the stator leading edge, $V_2$ is the velocity magnitude at midspan at the stator trailing edge, and $V_{1\theta}$ and $V_{2\theta}$ are the tangential velocities at the leading and trailing edge respectively. The time-averaged values of diffusion factor were calculated and are given in Table 5.6. For a given loading the diffusion factor is essentially constant regardless of rotor bow shock strength. At decreased, nominal, and increased loading the average diffusion factor for each loading is 0.3362, 0.4015, and 0.4287 respectively. As is expected, diffusion factor increases with swirler stagger angle. The increase from decreased to nominal is larger than from nominal to increased (0.065 compared to 0.027). It should be mentioned that the design diffusion factor was 0.45.

$$DF = \left(1 - \frac{V_2}{V_1}\right) + \frac{V_{1\theta} - V_{2\theta}}{2\sigma V_1}$$ (5.2)
Table 5.6: Time-Averaged Diffusion Factor for Each Configuration.

<table>
<thead>
<tr>
<th></th>
<th>−3.0°</th>
<th>0.0°</th>
<th>+1.5°</th>
</tr>
</thead>
<tbody>
<tr>
<td>Close</td>
<td>0.3262</td>
<td>0.3868</td>
<td>0.4200</td>
</tr>
<tr>
<td>Mid</td>
<td>0.3445</td>
<td>0.4124</td>
<td>0.4374</td>
</tr>
<tr>
<td>Far</td>
<td>0.3378</td>
<td>0.4054</td>
<td>—</td>
</tr>
<tr>
<td>Average</td>
<td>0.3362</td>
<td>0.4015</td>
<td>0.4287</td>
</tr>
</tbody>
</table>

In order to ensure that diffusion factor was a good measure of actual blade loading the time-averaged static pressure distribution was integrated over the deswirler chord at midspan. The results are plotted against the measured values of diffusion factor at midspan in Figure 5.3. The linear fit to the data shows that from a design standpoint, diffusion factor is an adequate measure of the blade loading on the deswirler. There is less than 2% average error from what a linear fit to the data would predict, with the largest error occurring for far spacing at nominal loading (4%). This shows that diffusion factor captures blade loading sufficiently enough for this model.

The rotor bow shock strength has also been shown to be an important parameter in determining vortex circulation from previous research [13,14] and from this research. The parameter $\Delta p$ is a measure of the shock strength, and is a commonly used parameter in compressor design. This was also used in the model developed by Gorrell [13]. $\Delta p$ is defined as the static pressure jump across the shock near the trailing edge of the stator, or the difference between the maximum and minimum static pressures across the shock, and $\bar{p}$ is defined as the average of the maximum and minimum static pressures. The shock strength was calculated at 20% pitch above the blade of interest at the deswirler trailing edge (100% chord), as shown in Figure 5.4. A typical static pressure profile of the rotor bow shock at the deswirler trailing edge is given in Figure 5.5. For each vortex measured in this analysis, the shock that caused the vortex to be shed was tracked through the time-accurate results. This shock strength was calculated, and the average values for each simulation are given in Table 5.7. As was expected, shock strength is essentially constant for each axial spacing. A wide range of shock strengths is observed in this analysis. For far spacing the average shock strength is 0.187, mid spacing is 0.373, and close is 0.625. Shock strength increases with decreased axial spacing. From far to mid axial spacing
the average shock strength increased by 99% and from mid to close spacing the average shock strength increased by 67%.

Table 5.7: Time-Averaged Shock Strength, $\frac{\Delta p}{p}$, for Each Configuration.

<table>
<thead>
<tr>
<th></th>
<th>$-3.0^\circ$</th>
<th>$0.0^\circ$</th>
<th>$+1.5^\circ$</th>
<th>Average</th>
</tr>
</thead>
<tbody>
<tr>
<td>Close</td>
<td>0.6149</td>
<td>0.5992</td>
<td>0.6596</td>
<td>0.625</td>
</tr>
<tr>
<td>Mid</td>
<td>0.3533</td>
<td>0.3884</td>
<td>0.3771</td>
<td>0.373</td>
</tr>
<tr>
<td>Far</td>
<td>0.1763</td>
<td>0.1983</td>
<td>—</td>
<td>0.187</td>
</tr>
</tbody>
</table>

Various other parameters were thought to be important, such as the shock angle, freestream velocity, trailing edge thickness, solidity, the blade passing frequency, and the Reynolds number. For these simulations the trailing edge thickness, solidity, and blade passing frequency were assumed constant. The shock angle changed negligibly between simulations, so it was
Figure 5.4: A Sample Shock Extraction Line, at 20% Pitch Above the TE

also assumed constant. Reynolds number and freestream velocity change negligibly between simulations as well. Multiple operating conditions were not simulated in this analysis, since it was beyond the scope of this research.

Since diffusion factor and shock strength are both dimensionless and circulation has units of \([m^2/s]\), a characteristic length and velocity were needed to develop a \(\Pi\) term. Contemporary transonic compressor stators and rotors have sharp leading and trailing edges, thus the trailing edge thickness was thought to be a poor choice for a characteristic length. Epstein [23] noted that the wake displacement thickness seemed to be a better representation of the blade trailing edge thickness for airfoils with sharp trailing edges. In this analysis the trailing edge wake displacement thickness was used as a characteristic length. The wake displacement thickness was calculated 2% chord downstream from the stator trailing edge from Equation 5.3, and the results for each simulation are given in Table 5.8. The wake displacement thickness changed negligibly for varying axial spacings, and the main difference is observed at varying loading. The results show that the wake displacement thickness increases with stator loading. On average \(\delta^*\) increased by 32% from decreased to nominal loading, and 25% from nominal to increased
A characteristic velocity was also necessary for the development of a $\Pi$ term. It was noted previously that the large vortices seemed to remain at the trailing edge longer than the normal vortices before rolling-off. The low-velocity region at the trailing edge caused the vortices to form over a longer time, allowing more high-strength, negative vorticity from the pressure side boundary layer to roll-up in the vortex before the vortex was pushed off the stator.

\[
\delta^* = \int_{-\infty}^{\infty} \left(1 - \frac{u}{u_\infty}\right) \, d\gamma = \left(\int_{-\infty}^{0} \left(1 - \frac{u}{u_\infty}\right) \, d\gamma\right)_{SS} + \left(\int_{0}^{\infty} \left(1 - \frac{u}{u_\infty}\right) \, d\gamma\right)_{PS} \tag{5.3}
\]
Table 5.8: Time-Averaged Wake Displacement Thickness, $\delta^*$, for Each Configuration (in mm).

<table>
<thead>
<tr>
<th></th>
<th>$-3.0^\circ$</th>
<th>$0.0^\circ$</th>
<th>$+1.5^\circ$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Close</td>
<td>2.124</td>
<td>3.163</td>
<td>3.542</td>
</tr>
<tr>
<td>Mid</td>
<td>2.401</td>
<td>2.805</td>
<td>3.982</td>
</tr>
<tr>
<td>Far</td>
<td>2.355</td>
<td>3.093</td>
<td>—</td>
</tr>
</tbody>
</table>

trailing edge. For this reason the average suction surface boundary layer velocity, $\bar{U}_{ss}$ was used as the characteristic velocity to complete the final $\Pi$ term. This was computed by averaging the axial velocity across the stator wake, 2% downstream of the trailing edge. The results of this are given in Table 5.9. It is expected that the trends observed in the suction surface boundary layer velocity are the inverse of the wake displacement thickness, since a larger wake defect would result in a larger $\delta^*$ and a lower $\bar{U}_{ss}$. This is indeed true. For the most part the suction side boundary layer velocity decreases with increased loading, but only slightly. From decreased to nominal loading $\bar{U}_{ss}$ decreases by 4% and from nominal to increased loading $\bar{U}_{ss}$ decreased by 3%.

Table 5.9: Time-Averaged Suction Surface Boundary Layer Velocity for Each Configuration (in m/s).

<table>
<thead>
<tr>
<th></th>
<th>$-3.0^\circ$</th>
<th>$0.0^\circ$</th>
<th>$+1.5^\circ$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Close</td>
<td>165.4</td>
<td>153.8</td>
<td>151.9</td>
</tr>
<tr>
<td>Mid</td>
<td>162.8</td>
<td>157.3</td>
<td>147.4</td>
</tr>
<tr>
<td>Far</td>
<td>154.4</td>
<td>151.9</td>
<td>—</td>
</tr>
</tbody>
</table>

Using these parameters it was determined that vortex circulation was a function of the wake displacement thickness, the suction surface boundary layer velocity, diffusion factor, and the rotor bow shock strength, as shown in Equation 5.4.

$$
\Pi_1 = f(\Pi_2, \Pi_3)
$$

$$
\frac{\Gamma}{(\delta^*)\bar{U}_{ss}} = f\left(DF, \frac{\Delta p}{\bar{p}}\right)
$$

(5.4)
5.3 Model Development

The functional relationship of Equation 5.4 was determined by trying combinations of multiplications, divisions, exponents, and constants. Further analysis showed that “large” vortex strength appeared to be linear with blade loading at mid spacing, where blade loading was defined as the integration of the pressure coefficient over the stator chord. The “normal” vortices at mid spacing appeared to remain constant with blade loading. These results are represented in Figure 5.2. For this reason it was determined that the functional relationship between diffusion factor and circulation should be linear.

In order to determine the relationship between shock strength and circulation, plots of circulation against shock strength were created for a constant incidence (blade loading). These are shown in Figure 5.6. It was observed that this relationship was essentially linear as well for a given loading. This resulted in the functional relationship for predicted circulation as shown in Equation 5.5, where $C_1$ and $C_2$ are constants to be determined by the fit to the data.
\[ \Gamma = C_1 \delta^* U_{ss} DF \left( \frac{\Delta \rho}{\bar{p}} \right) + C_2 \]  \hfill (5.5)

Constants were determined in order to fit the “large” vortex strength. The results were \( C_1 = 3.41 \) and \( C_2 = 0.55 \). This resulted in a very good fit to the “large” vortex data. The model resulted in an average error of 4.8%, with the largest error of 8.6% for mid spacing at nominal loading. The predicted circulation values are plotted against the CFD measured circulation values for the “large” values in Figure 5.7. The \( R^2 \) value of this linear fit is 0.89.

Since all of the vortices, both “normal” and “large”, were shock-induced, constants were developed to fit the averaged data shown in Table 5.3. The value of \( C_1 \) remained 3.41, and \( C_2 = 0.38 \). This resulted in an average error of 5.3%, with the largest error of 10% occurring for mid spacing at nominal loading. The predicted circulation values are plotted against the CFD measured average circulation values in Figure 5.8. The \( R^2 \) value of this linear fit is 0.91.
The model is surprisingly simple, but captures all the flow phenomena in these simulations. The vortex circulation can be predicted from various design parameters. The diffusion factor, wake displacement thickness, and average boundary layer velocity can be calculated from steady state simulations, and the rotor bow shock strength could be calculated from compressible flow theory or a steady state simulation. All of these parameters can be found easily without the use of costly high-fidelity CFD simulations.

If a different rotor bow shock angle were present in the flow, then this would need to be accounted for. This was not investigated in this analysis, but it is assumed that the result would be similar to the Langford and Gorrell models, given previously in Equations 2.1 and 2.2, in that the model would be multiplied by a $\cos\theta$ or $\tan\phi$. This is not investigated in this analysis, but may be worthy of future research in order to develop a more general model.

There is some variation in the data that can be attributed to variations in the parameters of interest. The shock strength varied by 8-17% for all the measured shocks at a given axial spacing. This is most likely due to the variation in the deswirler wake from an adjacent blade.
As the rotor bow shock passes through the deswirler wake it is affected, but an investigation of this phenomenon is beyond the scope of this analysis. For this reason the average shock strength for a given simulation was used. Likewise, the diffusion factor varied for the same stagger angle with different axial spacings by about 3%. This could be due in part to the effects of the rotor bow shock impinging on the stator. With a larger rotor bow shock the unsteady pressure loading is stronger, and affects the loading on the deswirler. Considering this variation, the model predicts the shock-induced vortex strength very accurately. It should be noted that an actual rotor geometry will have variations which will change the shock strength slightly. The same is true for the diffusion factor on stator vanes.
CHAPTER 6. DISCUSSION OF RESULTS

This chapter contains a discussion of the results presented in the previous chapters. The time-averaged and time-accurate results are both discussed. The results showing the suction surface boundary layer behavior, the formation of the low-velocity region, and some of the vortex shedding were qualitative in nature but are discussed here as they apply to the strength of the shed shock-induced vortices. The quantitative circulation results are also discussed, with a note on design implications.

Vortex strength can be related to entropy generation through Crocco's Theorem: stronger vortices contain stronger velocity gradients, which generate more entropy as they dissipate, resulting in more loss. This helps explain the experimentally observed decrease in stage performance at increased loading, since larger and stronger vortices are generated at decreased axial spacing and increased stator loading. These same trends were also observed for these numerical simulations.

This vortex circulation increase can result in increased loss downstream as well. Though the interaction of these stator-shed shock-induced vortices with the rotor bow shock, or their propagation through the rotor passage is not investigated in this research, it has been shown that stronger vortices cause additional loss downstream. Nolan [21] showed that the upstream stator-shed vortices influence the stagnation pressure distribution that encases the rotor. This affects the loss generation in the rotor boundary layer. List [18] also showed that the mean vortex trajectory through the rotor passage has a direct impact on rotor performance.

The magnitude of the pitchwise vortex radius has a direct effect on blockage and losses. Reynolds [16] showed that the pitchwise vortex radius grows with shock strength. A larger vortex will cause streamlines to deviate away from axial to go around the vortex and this process generates additional loss. For highly loaded, high Mach number turbomachines this aerody-
namic blockage affects the radial distribution of flow, flow swallowing capability of the stage, and flow angles entering and exiting a blade row [36].

The BRI rig was designed such that stator loading could be changed by adjusting the stagger angle of the swirler which changed the incidence to the deswirler and consequently the suction side boundary layer thickness. Incidence changes alone have an effect on boundary layer development. It has been shown that upstream propagating pressure waves from a rotor bow shock exacerbate boundary layer response to changes in incidence. The complex unsteady flow field has a significant effect on the boundary layer development in the stator passage which in turn affects the vortex shedding at the stator trailing edge. Such unsteady interactions must be accounted for to accurately predict the design and off-design performance of highly loaded, transonic fans and compressors. In order to do so these discussed parameters must be correctly modeled.

It is well known that a diffusion factor of 0.60-0.65 corresponds to flow separation on an airfoil. Though a transition or separation model are not employed in these simulations it appears that the suction surface separates periodically, but it cannot be known with certainty. Low-velocity, high-vorticity regions are seen to form far upstream on the deswirler suction surface and propagate downstream. From the analysis of the velocity contours at midspan it appears that these are some sort of boundary layer separation. If this is indeed the case, then separation is occurring at a much lower diffusion factor than 0.6. Diffusion factor for these simulations is between 0.32 and 0.44. This means that the impacting rotor bow shock can cause flow separation at a much lower diffusion factor than steady state solutions have shown.

While the author is unaware of previous research investigating the effects of propagating upstream pressure disturbances on stator boundary layer characteristics in high-speed turbo-machines, there has been research investigating the effects of upstream wakes on boundary layer development. There are some interesting similarities. Walraevens and Cumpsty [56] conducted experiments showing increases in local incidence moved the stagnation point toward the suction surface. This was observed in this analysis when the pressure wave which originated as a rotor bow shock propagated to the leading edge of the stator, changing the leading edge pressure gradient.
Wheeler and Miller [57] showed that fluctuations in momentum thickness at a stator leading edge propagated as thickened boundary layer structures over the blade surface at approximately 65% of the freestream velocity. They also observed that fluctuations close to the leading edge affected the boundary layer all the way to the trailing edge. It has been similarly observed that as a separation bubble forms near the stator leading edge a low velocity region grows and convects downstream. The low velocity region increases in size each time it passes through a pressure gradient generated from the rotor bow shock. When a shock-induced vortex forms in this low-velocity region at the stator trailing edge it is much stronger than it would be otherwise.

The experimental work performed by Langford [14] used a shock tube in a cascade facility to simulate the passing of a rotor. The data from this analysis did not match the model developed by Langford. Possible reasons for this are the changing diffusion factor and geometry. It should also be noted that it was not possible for the shock tube experiment to produce “large” vortices, as this seems to only occur after repeated shock impingement. This analysis has shown the importance of accounting for the periodic behavior of the rotor bow shock passing, as these effects appear to be more complex than a discrete shock impingement event.

The unsteady pressure loading variation is an important thing to consider from a structural standpoint. The desire to decrease engine weight has driven the structural design of vanes and blades. Engine designers are very concerned with high-cycle fatigue (HCF), which causes the vanes and blades to fail due to the unsteady effects in turbomachines. In these simulations the effects of the varying pressure loading on the stator structure has not been studied, but designers are very interested in understanding this problem in order to increase the lifetime of the engines. Blade-row interactions play a very prominent role in high-cycle fatigue.

For these simulations the developed model predicts the circulation of a shock-induced vortex very accurately. A wide range of vortex circulations existed for each configuration, but the averaged values will help give compressor designers insight into the flow physics and unsteady phenomena. Contemporary design tools do not directly account for unsteady effects, such as blade-row interactions. This model will allow designers to take into account the unsteady vortex shedding that takes place at a stator trailing edge upstream of a transonic rotor early on in the
design process. This will allow for a more accurate estimate of blockage, radial transport and entropy generation, and could lead to improved stage efficiency and pressure rise.

The model that predicts the circulation of the “large” vortices gives engineers understanding into how stators operating near stall might behave. These vortices form in a large, low-velocity region that has developed far upstream on the stator suction surface and has propagated downstream along the stator chord to the trailing edge. Off-design performance is very difficult to simulate, but these simulations give insight into how a slightly changed stator incidence (1.5 – 3.0°) performs due to the change in shed vortex strength and the suction surface boundary layer behavior. At slightly higher loadings large changes in vortex strength are experienced, as well as significantly different suction surface boundary layers. As diffusion factor increased by 6.5% from decreased to nominal loading average circulation increased by 12%. As diffusion factor increased from nominal to increased loading by 2.7% the average circulation increased by 14%. Circulation is also affected by shock strength, but not as much as diffusion factor. As shock strength increased from far to mid spacing by 99% the average circulation increased by 29%. As the shock strength increased from mid to close spacing by 67% the average circulation increased by 20%. It is shown by this analysis that these parameters are very important to stage performance, and should be taken into account during the design process.

6.1 Recommendations for Future Work

Future work on the BRI rig may include a variety of analyses. An investigation of vortex shedding at other radial locations may yield interesting results. The current results were all taken at midspan, but the majority of losses occur in the tip region due to the clearance flows. Although this is a separate phenomenon, this may be an area worthy of future investigation. A further investigation of vortex shedding in transonic axial compressors may include the radial variation of the vortex strength. The data has been generated, and needs only to be extracted and analyzed. Though it was discussed that larger and stronger vortices cause more entropy generation, these effects were not quantified in this analysis. A further analysis could include an investigation of the loss-production due to vortex shedding. Previous studies showed that the mean vortex trajectory through the rotor passage affected rotor performance. This was beyond the scope of this analysis, but may be something to investigate in the future, especially the radial
variation of these effects. It was noted that the model should probably include the shock angle, but the shock angle was constant for these simulations. Future work should include a varying shock angle. These simulations were also all performed at peak efficiency, but an investigation of how the stators perform at off-design engine operating conditions may yield further insight. Simulations performed near stall or near choke may reveal other important unsteady blade-row interaction effects.
CHAPTER 7. CONCLUSION

Multiple high-fidelity, time-accurate CFD simulations have been performed at three stator/rotor axial spacings and three stator loadings to investigate the effects of these parameters on shock-induced vortex shedding in a transonic axial compressor. Quarter annulus simulations of a one and a half stage compressor were performed using the flow solver, TURBO.

Both the time-averaged and time-accurate CFD results match the experimental PIV and performance data well. The time-averaged simulation efficiencies consistently overpredict the experimental efficiencies by 5%, but the same trends observed in the experimental data are observed in the CFD data. It is common for CFD simulations to overpredict efficiency, so this is not a cause for concern, but it is, however, remarkable how closely the trends observed in the simulation efficiencies match the experimental trends. The change in efficiency between configurations for peak performance is matched. It is consistently observed that for a given axial spacing the trends are matched: as stator loading increases, stage efficiency decreases. For the most part, this same observation holds for a given axial spacing as well. As shock strength increases (decreasing axial spacing) for a given stator loading, stage efficiency decreases, except that mid spacing at decreased loading has been shown to be the most efficient. The same experimental trends in mass flow rates are observed at varying axial spacings for the CFD results as well. Mass flow rate decreases with increased stator loading.

The time-accurate results also match the experimental data very well. The average shock-induced vortex strength, or circulation, is slightly lower than the PIV results, but is within 8.3% of the experimental values on average. This shows that the simulations have reproduced the experiment very well, and are sufficiently validated. Conclusions about the stator loading, axial spacing, vortex strength, and the flow physics can now be drawn.

The effects of the rotor bow shock are felt far upstream, near the upstream stator leading edge. As the rotor bow shock impinges on the stator trailing edge, it reflects toward an adjacent
blade. These pressure waves propagate upstream and change the local pressure gradient at various locations on the suction surface of the adjacent blade. The pressure waves have also been shown to affect the leading edge pressure gradient of the stator, as they propagate up the pressure surface of the stator. A complex pressure gradient exists in the entire stator passage due to the passing of the rotor bow shock, and the subsequent interaction of the shock with the stator trailing edge.

A thick boundary layer or separation bubble on the suction surface of a stator has been shown to increase the circulation of shock-induced vortices formed on the trailing edge of the stator. This leads to the formation of a low-velocity region at the trailing edge of the suction surface of the stator. The low-velocity region originates far upstream on the stator suction surface, at 45% chord for decreased loading at midspan, and 20-30% chord for nominal loading. These regions of increased boundary layer growth propagate downstream, and grow in pitch as they pass through the various reflected pressure waves on the suction surface. A shock-induced vortex that forms in the low-velocity region at the trailing edge is much stronger than a vortex that forms normally. These “large” vortices are 1.7, 1.9, and 2.1 times stronger than the “normal” vortices for decreased, nominal, and increased loading respectively.

A stator with higher loading and a stronger impacting rotor bow shock can shed stronger and larger vortices than a stator with lower loading and lower shock strength. Circulation has been shown to increase with stator loading by 19% from decreased to nominal loading, and by 26% from nominal to increased loading. Circulation has also been shown to increase with decreasing axial spacing, from far to mid spacing by 21% and from mid to close spacing by 20%.

A model has been developed to predict shock-induced vortex strength given the diffusion factor, rotor bow shock strength at the stator trailing edge, wake displacement thickness, and average suction surface boundary layer velocity. The model predicts shed vortex strength well, with an $R^2$ value of 0.91 for the average shock-induced vortex circulation of each simulation. Circulation has been shown to increase linearly with shock strength and diffusion factor. This model will help give designers a tool that accounts for unsteady effects and predicts the strength of shock-induced vortices. Compressor designers will be able to predict shed vortex strength given a stator loading and rotor bow shock strength, and will allow designers to predict flow blockage and entropy generation due to shock-induced vortices. This will help designers
more accurately account for blade-row interactions early on in the design process, and produce more efficient engines that perform as designed.
REFERENCES


APPENDIX A. VORTICITY IMAGES

The same instantaneous data sets given in Figures 4.12-4.14 and 4.15-4.17 in Chapter 4 are reproduced here with vorticity lines and contours. Vorticity contours for mid spacing simulations for decreased, nominal, and increased loading are given here, for both “normal” and “large” vortices.
Figure A.1: Radial Vorticity Contours with Vorticity Lines Showing “Normal” Vortex Formation on Deswrler TE at Midspan for Mid Spacing at Decreased Loading. Compare to Figure 4.12.
Figure A.2: Radial Vorticity Contours with Vorticity Lines Showing “Normal” Vortex Formation on Deswirler TE at Midspan for Mid Spacing at Nominal Loading. Compare to Figure 4.13.
Figure A.3: Radial Vorticity Contours with Vorticity Lines Showing “Normal” Vortex Formation on Deswirler TE at Midspan for Mid Spacing at Increased Loading. Compare to Figure 4.14.
Figure A.4: Radial Vorticity Contours with Vorticity Lines Showing “Large” Vortex Formation on Deswirler TE at Midspan for Mid Spacing at Decreased Loading. Compare to Figure 4.15.
Figure A.5: Radial Vorticity Contours with Vorticity Lines Showing “Large” Vortex Formation on Deswirler TE at Midspan for Mid Spacing at Nominal Loading. Compare to Figure 4.16.
Figure A.6: Radial Vorticity Contours with Vorticity Lines Showing “Large” Vortex Formation on Deswirler TE at Midspan for Mid Spacing at Increased Loading. Compare to Figure 4.17.
APPENDIX B. GRID PARTITIONING

The partitioning information for each simulation performed is given here. As was mentioned in Chapter 3, partitioning was performed in order to yield the same average number of nodes for each partition.

Table B.1: Partitioning Information for Mid Spacing at -3.0° Stagger (Decreased Loading) Resulting in 348 Partitions.

<table>
<thead>
<tr>
<th></th>
<th>Axial</th>
<th>Radial</th>
<th>No. Blades</th>
<th>No. Blocks</th>
<th>Avg. Nodes/Block</th>
</tr>
</thead>
<tbody>
<tr>
<td>Swirler</td>
<td>5</td>
<td>2</td>
<td>8</td>
<td>80</td>
<td>438,000</td>
</tr>
<tr>
<td>Deswirler</td>
<td>8</td>
<td>2</td>
<td>8</td>
<td>128</td>
<td>450,000</td>
</tr>
<tr>
<td>Rotor</td>
<td>10</td>
<td>2</td>
<td>7</td>
<td>140</td>
<td>439,000</td>
</tr>
<tr>
<td><strong>Total</strong></td>
<td></td>
<td></td>
<td></td>
<td><strong>348</strong></td>
<td></td>
</tr>
</tbody>
</table>

Table B.2: Partitioning Information for Mid Spacing at 0.0° Stagger (Nominal Loading) Resulting in 522 Partitions. Note that Mid Spacing at +1.5° Stagger (Increased Loading) Results in the Same Partitions.

<table>
<thead>
<tr>
<th></th>
<th>Axial</th>
<th>Radial</th>
<th>No. Blades</th>
<th>No. Blocks</th>
<th>Avg. Nodes/Block</th>
</tr>
</thead>
<tbody>
<tr>
<td>Swirler</td>
<td>5</td>
<td>3</td>
<td>8</td>
<td>120</td>
<td>292,000</td>
</tr>
<tr>
<td>Deswirler</td>
<td>8</td>
<td>3</td>
<td>8</td>
<td>192</td>
<td>300,000</td>
</tr>
<tr>
<td>Rotor</td>
<td>10</td>
<td>3</td>
<td>7</td>
<td>210</td>
<td>293,000</td>
</tr>
<tr>
<td><strong>Total</strong></td>
<td></td>
<td></td>
<td></td>
<td><strong>522</strong></td>
<td></td>
</tr>
</tbody>
</table>
### Table B.3: Partitioning Information for Far Spacing at -3.0° Stagger (Decreased Loading) Resulting in 516 Partitions.

<table>
<thead>
<tr>
<th></th>
<th>Axial</th>
<th>Radial</th>
<th>No. Blades</th>
<th>No. Blocks</th>
<th>Avg. Nodes/Block</th>
</tr>
</thead>
<tbody>
<tr>
<td>Swirler</td>
<td>6</td>
<td>2</td>
<td>8</td>
<td>96</td>
<td>328,000</td>
</tr>
<tr>
<td>Deswirler</td>
<td>14</td>
<td>2</td>
<td>8</td>
<td>224</td>
<td>329,000</td>
</tr>
<tr>
<td>Rotor</td>
<td>14</td>
<td>2</td>
<td>7</td>
<td>196</td>
<td>313,000</td>
</tr>
<tr>
<td><strong>Total</strong></td>
<td></td>
<td></td>
<td></td>
<td>516</td>
<td></td>
</tr>
</tbody>
</table>

### Table B.4: Partitioning Information for Far Spacing at 0.0° Stagger (Nominal Loading) Resulting in 516 Partitions.

<table>
<thead>
<tr>
<th></th>
<th>Axial</th>
<th>Radial</th>
<th>No. Blades</th>
<th>No. Blocks</th>
<th>Avg. Nodes/Block</th>
</tr>
</thead>
<tbody>
<tr>
<td>Swirler</td>
<td>3</td>
<td>4</td>
<td>8</td>
<td>96</td>
<td>328,000</td>
</tr>
<tr>
<td>Deswirler</td>
<td>7</td>
<td>4</td>
<td>8</td>
<td>224</td>
<td>329,000</td>
</tr>
<tr>
<td>Rotor</td>
<td>7</td>
<td>4</td>
<td>7</td>
<td>196</td>
<td>313,000</td>
</tr>
<tr>
<td><strong>Total</strong></td>
<td></td>
<td></td>
<td></td>
<td>516</td>
<td></td>
</tr>
</tbody>
</table>

### Table B.5: Partitioning Information for Close Spacing at -3.0° Stagger (Decreased Loading) Resulting in 996 Partitions.

<table>
<thead>
<tr>
<th></th>
<th>Axial</th>
<th>Radial</th>
<th>No. Blades</th>
<th>No. Blocks</th>
<th>Avg. Nodes/Block</th>
</tr>
</thead>
<tbody>
<tr>
<td>Swirler</td>
<td>12</td>
<td>3</td>
<td>8</td>
<td>288</td>
<td>134,000</td>
</tr>
<tr>
<td>Deswirler</td>
<td>12</td>
<td>3</td>
<td>8</td>
<td>288</td>
<td>144,000</td>
</tr>
<tr>
<td>Rotor</td>
<td>20</td>
<td>3</td>
<td>7</td>
<td>420</td>
<td>146,000</td>
</tr>
<tr>
<td><strong>Total</strong></td>
<td></td>
<td></td>
<td></td>
<td>996</td>
<td></td>
</tr>
</tbody>
</table>

### Table B.6: Partitioning Information for Close Spacing at 0.0° Stagger (Nominal Loading) Resulting in 996 Partitions. Note that Close Spacing at +1.5° Stagger (Increased Loading) Results in the Same Partitions.

<table>
<thead>
<tr>
<th></th>
<th>Axial</th>
<th>Radial</th>
<th>No. Blades</th>
<th>No. Blocks</th>
<th>Avg. Nodes/Block</th>
</tr>
</thead>
<tbody>
<tr>
<td>Swirler</td>
<td>6</td>
<td>6</td>
<td>8</td>
<td>288</td>
<td>134,000</td>
</tr>
<tr>
<td>Deswirler</td>
<td>6</td>
<td>6</td>
<td>8</td>
<td>288</td>
<td>144,000</td>
</tr>
<tr>
<td>Rotor</td>
<td>10</td>
<td>6</td>
<td>7</td>
<td>420</td>
<td>146,000</td>
</tr>
<tr>
<td><strong>Total</strong></td>
<td></td>
<td></td>
<td></td>
<td>996</td>
<td></td>
</tr>
</tbody>
</table>
APPENDIX C. MASS FLOW RATE HISTORIES

The mass flow rate history for the simulations is given here. The full mass flow rate history for each simulation is given, followed by the details of the time-averaged and time-accurate data output histories. In Chapter 3 the mass flow rate history for a few of the simulations was given. This appendix contains the mass flow rate histories for all the simulations used in this analysis.

C.1 Mid, Decreased Loading

As was mentioned in Chapter 3, the low-frequency oscillation in the mass flow rates can also be observed in Figure C.1 from iterations 40,000 to 80,000. The amplitude of the oscillation did not decay, but the mass flow rates reached a periodic state, so the simulation was declared converged. The time-averaged data was generated for five periods of a quarter annulus (1.25 full revolutions) as can be seen in Figure C.2. The mass flow rates are all within 0.2-0.25 kg/s of the target mass flow. The time-accurate solution generation can be seen in Figure C.3. The inlet mass flow rate is lower than desired, but the area of interest for the time-accurate data is between the rotor and deswirler, and the mass flow rates for these blade rows are all within 0.2 kg/s of the target mass flow rate.

C.2 Mid, Nominal Loading

For mid spacing at nominal loading the full mass flow rate history is given in Figure C.4. An error was made during the initial stages of this simulation and the wrong exit mass flow rate was imposed. This was changed just before 50,000 iterations, as can be seen in Figure C.4, and the flow solver adjusted to the change without diverging. The time-averaged solution was generated for one full rotor revolution, and the history for those iterations is seen in Figure C.5. The amplitude of the oscillation is large at this point, and multiple time-averages were taken.
for this configuration. This time-average solution was considered the best, as all blade-to-blade variations were eliminated. The unsteady data was output for a quarter of a revolution, and the history is shown in Figure C.6. The mass flow rates are all within 0.2 kg/s of the target mass flow rate.

### C.3 Mid, Increased Loading

Figure C.7 shows the full mass flow rate history for mid spacing at increased loading. This simulation converged very rapidly compared to others. The amplitude of the low-frequency oscillation decayed down to 0.2 kg/s. The history for the time-averaged data output is given in Figure C.8, and was generated over a full rotor revolution. The history for the unsteady data output is given in Figure C.9. For both of these, the mass flow rates stay within 0.2 kg/s of the target mass flow rate.
C.4 Far, Decreased Loading

The full mass flow rate history for far spacing at decreased loading is given in Figure C.10. The simulation had excellent low-order convergence, as seen up to 38,000 iterations. The increase in the amplitude of the low-frequency oscillation is observed after 38,000 iterations. The amplitude never completely decayed, but was periodic, so it was considered converged. The mass flow rate history for the time-averaged data output (one full rotor revolution) is given in Figure C.11, and the unsteady in Figure C.12.

C.5 Far, Nominal Loading

The full mass flow rate history for far spacing at nominal loading is given in Figure C.13. Note that the solver order was increased at 45,000 iterations, resulting in a large increase in the amplitude of the low-frequency oscillation. The mass flow rates decayed slightly, but were considered converged due to the periodic behavior. The history for the time-averaged solution output is given in Figure C.14, and was generated for 3/4 of a full rotor revolution. The mass flow rates were all near the target mass flow rate for the solution generation. The mass flow rate
Figure C.3: Mass Flow Rate History for the Unsteady Solution Generation for Mid Spacing at Decreased Loading.

The mass flow rates were all within 0.15 kg/s of the target mass flow rate.

C.6 Close, Decreased Loading

Figure C.16 shows the first 115,000 iterations of the full mass flow rate history for close spacing at decreased loading. It is interesting to note that this simulation nearly diverged during the initial transients, but the solver was able to recover and reach a converged simulation. This, however, took many more iterations than the other simulations. Around 65,000 iterations the solver order was increased, which resulted in the increase in the amplitude of the low-frequency oscillation. The amplitude decayed slightly, but oddly, when the time step was decreased the amplitude decreased greatly (around 105,000 iterations in Figure C.16). The mass flow rate history for the time-averaged solution generation is give in Figure C.17, and was generated over 1.25 full rotor revolutions. The unsteady solution generation history is given in Figure C.18.
Figure C.4: Mass Flow Rate History for Mid Spacing at Nominal Loading.

Figure C.5: Mass Flow Rate History for the Time-Averaged Solution Generation for Mid Spacing at Nominal Loading.
Figure C.6: Mass Flow Rate History for the Unsteady Solution Generation for Mid Spacing at Nominal Loading.

Figure C.7: Mass Flow Rate History for Mid Spacing at Increased Loading.
Figure C.8: Mass Flow Rate History for the Time-Averaged Solution Generation for Mid Spacing at Increased Loading.

Figure C.9: Mass Flow Rate History for the Unsteady Solution Generation for Mid Spacing at Increased Loading.
Figure C.10: Mass Flow Rate History for Far Spacing at Decreased Loading.

Figure C.11: Mass Flow Rate History for the Time-Averaged Solution Generation for Far Spacing at Decreased Loading.
Figure C.12: Mass Flow Rate History for the Unsteady Solution Generation for Far Spacing at Decreased Loading.

Figure C.13: Mass Flow Rate History for Far Spacing at Nominal Loading.
Figure C.14: Mass Flow Rate History for the Time-Averaged Solution Generation for Far Spacing at Nominal Loading.

Figure C.15: Mass Flow Rate History for the Unsteady Solution Generation for Far Spacing at Nominal Loading.
Figure C.16: Mass Flow Rate History for Close Spacing at Decreased Loading.

Figure C.17: Mass Flow Rate History for the Time-Averaged solution Generation for Close Spacing at Decreased Loading.
Figure C.18: Mass Flow Rate History for the Unsteady Solution Generation for Close Spacing at Decreased Loading.
APPENDIX D. TURBO INPUT FILES

This appendix includes the input files used for the TURBO simulations performed at mid spacing. The files included in this section are as follows. ‘input00’ details the overall simulation parameters. ‘input01’ has parameters specifically for blade row 1 (swirler), ‘input02’ has similar parameters specifically for blade row 2 (deswirler), and ‘input03’ contains the parameters specific to blade row 3 (rotor). For the sake of brevity, the input files for all of the simulations are not given here. The only differences between close, mid, and far at each loading are the target mass flow rates. These are given for each simulation in Table 3.4.

D.1 Input Files

D.1.1 input00

&PARAMETERS
num_blade_rows=3
debug=F
gofast=T/
&SOLUTION_PARAMETERS
num_printouts=1
num_iter_per_printout=2240
jacobian_update=1
resid_print=1
max_num_subiter=6
min_residual=-1.2000
num_iter_zero_grad_bc=0
num_iter_first_order=0
num_iter_infreq_jacobian_update=0
num_iter_inviscid=0
num_iter_without_fluxfix=0
symmetry_factor=4
num_iter_restart_write=320
slip_bc_type=1
freeze_jacobian=0
num_sgs_iter=3
solution_type=2
turbulence_model=5
temporal_accuracy=2
spatial_accuracy=3
limiter_flag=1
spatial_scheme=0
wall_bc_limiter_flag=0
wall_bc_first_order_i=1
wall_bc_first_order_j=1
wall_bc_first_order_k=1
trap_negative=F
solution_correct_method=2
pressure_clip=0.02800000/

&SLIDING_BC
use_conserve_bc=0/

&INITIAL_CONDITION
initialize_solution=4
freestream_mach_num=0.5300
tangential_angle=0.0000
radial_angle=0.0000
thru_ic_flag =1/

&REFERENCE_CONDITIONS
ref_pressure=101325.0000
ref_temperature=288.1500
ref_velocity=287.567
ref_gamma=1.401290
ref_length=1.0000
gamma_table=1.40129 1.40129
temp_gam_table=288.15 388.15
gamref_t1=1.40129

&ke_MODEL_PARAMETERS
kemdl_input_type=0
kemdl_init_option=0
inlet_turbulence_intensity=0.0200
inlet_eddy_viscosity=30.0000
spatial_accuracy_2eq=3
temporal_accuracy_2eq=2
limiter_flag_2eq=1
use_pgrad_term=F
use_emut_wall_damping=T
max_num_subiter_2eq=1
jacobian_update_2eq=1
num_sgs_iter_2eq=1
cmu_clip_max=1.0000
dmut_clip_max=1.0000/

&TIME_SHIFT_BC
  use_time_shift_bc=0
  time_shift_bc_factor=0.500
  initialize_time_shift_in_TURBO=F/

&INLET_BC
  inlet_bc_type=-2
  num_blade_us=0
  omega_blade_us=0.0000
  num_tangential_cells_us=1/

&EXIT_BC
  exit_bc_type=23
  back_pressure=122850
  num_blade_ds=49
  omega_blade_ds=0.0
  num_tangential_cells_ds=1/

&FLUTTER
  flutter_grid_type=0
  flutter_freq=0.0000
  flutter_phase_angle=0.0000/

&TIME_STEP
  cfl=0.0000
  use_local_time_step=0
  num_time_steps_per_period=2240
  omega_ts=-16487.025
  num_blds_ts=4/

&OUTPUT
  output_format=3
  num_iter_per_soln_dump=20
  num_soln_per_flow_file=1
  num_parallel_writes=9999
  itime_start_anim=999999
  tmavg_output=F/

&INLET_PROFILE
  span = 0 0.05 0.25 0.5 0.75 0.95 1
  total_pressure = 86614.8 101325 101325 101325 101325 101325 87588.8
  total_temperature = 7*288.15
  tangential_angle = 7*0
  radial_angle = 7*0/

&EXIT_PROFILE
  span = 0.0 1.0
  static_pressure = 80000
  exit_mass_flow = 14.892/
D.1.2  input01

&BLADE_ROW_PARAMETERS
omega_bld = 0.0
num_blades = 32
suction_surface = 1
use_wall_func_i = 1
use_wall_func_j = 1
use_wall_func_k = 1
transition = 0.1
blade_row_ksym = 4
num_time_steps Stored = -1
num_adjacent_blades = 32
x_start_case_rotation = 9999.
x_end_case_rotation = 9999.
x_start_hub_rotation = -9999.
x_end_hub_rotation = -9999.
/

D.1.3  input02

&BLADE_ROW_PARAMETERS
omega_bld = 0.0
num_blades = 32
suction_surface = 2
use_wall_func_i = 1
use_wall_func_j = 1
use_wall_func_k = 1
transition = 0.1
blade_row_ksym = 4
num_time_steps Stored = -1
num_adjacent_blades = 4
x_start_case_rotation = 9999.
x_end_case_rotation = 9999.
x_start_hub_rotation = -9999.
x_end_hub_rotation = -9999.
/

D.1.4  input03

&BLADE_ROW_PARAMETERS
omega_bld = -16487.025
num_blades = 28
suction_surface = 1
use_wall_func_i = 1
use_wall_func_j = 1
use_wall_func_k = 1
transition = 0.1
blade_row_ksym = 4
num_time_steps_stored = -1
num_adjacent_blades = 32
x_start_case_rotation = 9999.
x_end_case_rotation = 9999.
x_start_hub_rotation = -0.0031750007
x_end_hub_rotation = 0.06032501
/