EFFECTS OF STATOR PRESSURE FIELD ON
UPSTREAM ROTOR PERFORMANCE

by

Martin Bowyer Graf

B.S., Mechanical Engineering, University of Miami (Florida), 1990
S.M., Mechanical Engineering, Massachusetts Institute of Technology, 1993
S.M., Aeronautics and Astronautics, Massachusetts Institute of Technology, 1996

Submitted to the Department of Mechanical Engineering
in Partial Fulfillment of the Requirements for the
Degree of

DOCTOR OF PHILOSOPHY

at the

Massachusetts Institute of Technology

June 1996

© Massachusetts Institute of Technology, 1996. All rights reserved.

Signature of Author

Department of Mechanical Engineering
May 3, 1996

Certified by

Edward M. Greitzer
H.N. Slater Professor of Aeronautics and Astronautics and Director, GTL
Thesis Supervisor

Certified by

Wai K. Cheng
Associate Professor of Mechanical Engineering
Committee Chairman

Certified by

J.B. Kerrebrock
R.C. MacLaurin Professor of Aeronautics and Astronautics

Certified by

Richard L. and Dorothy M. Hayman Professor of Mechanical Engineering
and Professor of Jet Propulsion, Emeritus, California Institute of Technology

Certified by

C.S. Tan
Research Engineer

Accepted by

Massachusetts Institute of Technology
Professor Ain A. Sonin
Chairman, Departmental Graduate Committee

JUN 27 1996
EFFECTS OF STATOR PRESSURE FIELD ON
UPSTREAM ROTOR PERFORMANCE

by

Martin Bowyer Graf

Submitted to the Department of Mechanical Engineering
on May 4, 1996 in Partial Fulfillment of the
Requirements for the Degree of Doctor of Philosophy in
Mechanical Engineering

ABSTRACT

The effects of stator pressure field on upstream rotor performance in a high pressure compressor stage have been assessed using three-dimensional steady and time-accurate Reynolds-averaged Navier-Stokes computations. Emphasis was placed on: (1) determining the dominant features of the unsteady flow arising from interaction of the rotor with the stator pressure field, and (2) quantifying the overall effects on time averaged loss, blockage, and pressure rise. The time averaged results showed 20 to 40% increases in overall rotor loss and 10 to 50% decreases in tip clearance loss compared to an isolated rotor. The differences became larger as the stage pressure rise and the amplitude of the unsteady back pressure variations was increased.

Motions of the tip leakage vortex on the order of the blade pitch were observed at the rotor exit in all the unsteady flow simulations. The period of the motion scaled with the rotor flow-through time rather than with the stator passing (i.e. the forcing period). In terms of time averaged quantities the movement of the tip vortex enhanced mixing giving an endwall loss which was within 20% of that computed assuming fully mixed out conditions.

Three steady flow approximations for the rotor-stator interaction were quantitatively assessed: a simple axisymmetric representation of the stator pressure field, an inter-blade row averaging plane method, and a technique which incorporated the deterministic stresses and bodyforces associated with stator flow field. The differences between the steady and unsteady flow predictions of overall rotor loss, tip region loss, and endwall blockage ranged from 5 to 50% of the time averaged values. However, the steady flow models overall rotor pressure rise and flow capacity values within 5% of the time averages obtained from the unsteady flow simulations.

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

Throughout the course of this project many individuals have provided me with motivation, guidance, and assistance. I wish to take this opportunity to thank all those who contributed, either directly or indirectly, to the completion of this work.

First, I would like to express my gratitude to my advisor, Prof. E. M. Greitzer, for his support, constructive criticism, and friendly encouragement throughout this project. Thanks also to Dr. C. S. Tan for his technical advice and the entertaining non-technical discussions, to Prof. F. E. Marble for his insightful questions and encouraging words, to Prof. J. L. Kerrebrock for many helpful comments, and to Prof. W. K. Cheng for his friendly participation as chairman of my doctoral committee.

This work would not have been possible without the support and assistance of several individuals at Pratt & Whitney in East Hartford, CT. My thanks to Dr. O. P. Sharma for his friendship, innumerable technical discussions, and the confidence he showed in my abilities. Thanks also to Mr. L. L. Coons for supporting my technical efforts and making this M.I.T.-P&W project possible. Many thanks to Mr. J. D. Buteau, Dr. R-H. R. Ni, and Dr. T. Shang for their assistance in performing the NiSTAR calculations. Similarly, many thanks to Mr. A. J. Gleixner, Dr. C. M. Rhie, and Mr. R. M. Zacharias for their assistance in executing the NASTAR calculations. Thanks also to Mr. S. Connolly for assistance in creating animations of the unsteady flow simulations.

Thanks to the past and present students of the GTL for their friendship and support, both of which have been greatly appreciated.

To my parents Robert and Leeda, and my sister Michelle, thanks for your constant encouragement, love, and advice. The support you have given me has kept me motivated throughout my years of graduate study, and I can never thank you enough.

Finally, special thanks to Diane. For all you have done to help me finish, you have my respect, my gratitude, and my love. Thanks for seeing me through.

The Air Force Office of Scientific Research provided funding for the author through the Air Force Research in Aero Propulsion Technology (AFRAPT) Fellowship program, Dr. J. McMichael, technical monitor. Support was also provided by the Pratt & Whitney Division of United Technologies Corporation, Mr. R. S. Mazzawy and Dr. O. P. Sharma technical monitors. Both sources are gratefully acknowledged.
Table of Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Abstract</td>
<td>2</td>
</tr>
<tr>
<td>Acknowledgments</td>
<td>3</td>
</tr>
<tr>
<td>List of Figures</td>
<td>7</td>
</tr>
<tr>
<td>List of Tables</td>
<td>11</td>
</tr>
<tr>
<td>Nomenclature</td>
<td>12</td>
</tr>
<tr>
<td>1. Introduction</td>
<td></td>
</tr>
<tr>
<td>1.1 Background and Motivation</td>
<td>16</td>
</tr>
<tr>
<td>1.2 Review of Previous Work</td>
<td>17</td>
</tr>
<tr>
<td>1.2.1 Tip Clearance Flows</td>
<td>18</td>
</tr>
<tr>
<td>1.2.2 Unsteady Flows: Rotor-Stator Interaction</td>
<td>21</td>
</tr>
<tr>
<td>1.3 Problem Statement and Research Objectives</td>
<td>24</td>
</tr>
<tr>
<td>1.4 Contributions of Thesis</td>
<td>25</td>
</tr>
<tr>
<td>1.5 Outline of Thesis</td>
<td>25</td>
</tr>
<tr>
<td>2. Background and Approach</td>
<td></td>
</tr>
<tr>
<td>2.1 Introduction</td>
<td>30</td>
</tr>
<tr>
<td>2.2 Approach: Computational Fluid Dynamics</td>
<td>30</td>
</tr>
<tr>
<td>2.3 Rotor and Stator Design Parameters</td>
<td>32</td>
</tr>
<tr>
<td>2.4 Description of Computational Fluid Dynamics Codes</td>
<td>35</td>
</tr>
<tr>
<td>2.4.1 NiSTAR</td>
<td>35</td>
</tr>
<tr>
<td>2.4.2 NASTAR</td>
<td>38</td>
</tr>
<tr>
<td>2.5 Code Validation</td>
<td>42</td>
</tr>
<tr>
<td>3. Steady Flow Simulations</td>
<td></td>
</tr>
<tr>
<td>3.1 Introduction</td>
<td>50</td>
</tr>
</tbody>
</table>
3.2 Rotor Flow Field ........................................... 51
  3.2.1 Rotor Performance Characteristics .................. 51
  3.2.2 Effect of Computational Grid Resolution .............. 53
  3.2.3 Aerodynamic Losses .................................. 55
  3.2.4 Aerodynamic Flow Blockage ......................... 58

3.3 Stator Flow Field ......................................... 60

3.4 Overall Stage Performance ............................... 62

3.5 Summary and Conclusion ................................ 63

4. Baseline Unsteady Flow Simulations

4.1 Introduction .............................................. 72

4.2 Rotor Flow Behavior ...................................... 73
  4.2.1 Fluid Dynamic Features ............................... 76
  4.2.2 Time Averaging ...................................... 81

4.3 Rotor Performance Characteristics ...................... 84
  4.3.1 Pressure Rise Capability ............................... 84
  4.3.2 Rotor with Axisymmetric Exit Condition .............. 85
  4.3.3 Aerodynamic Losses .................................. 88
  4.3.4 Aerodynamic Endwall Blockage ......................... 95

4.4 Observations from Unsteady Flow Simulations .......... 97

4.5 Questions to be Addressed by Sensitivity Study ........ 98

5. Sensitivity Study on Unsteady Flow Effects

5.1 Introduction .............................................. 114

5.2 Rotor Flow Behavior ...................................... 115

5.3 Rotor Performance Characteristics ...................... 118
  5.3.1 Overall Pressure Rise and Loss ....................... 119
  5.3.2 Effect of Downstream Stator Radial Design .......... 120
  5.3.3 Effect of Rotor-Stator Axial Spacing ................. 122
5.3.4 Effect of Rotor-Stator Blade Count Ratio .... 124
5.4 Rotor Response to Circumferentially Specified Exit Condition .... 126
5.5 Behavior of an Isolated Rotor ................. 127
5.6 Conclusions from the Sensitivity Study ............. 130

6. Conclusions

6.1 Summary and Conclusions ................. 141
6.2 Implications of Present Work ................. 144
6.3 Recommendations for Future Work ............. 144

References and Bibliography ............. 146

APPENDICES

(A) Summary of CFD Calculations Executed

A.1 Steady Flow ................. 152
A.2 Unsteady Flow ................. 153

(B) Stator Flow Behavior

B.1 Introduction ................. 154
B.2 Fluid Dynamic Features ................. 154
B.3 Stator Performance Characteristics ................. 156
  B.3.1 Pressure Rise Capability ................. 156
  B.3.2 Aerodynamic Losses ................. 157
B.4 Discussion of Sensitivity Study ................. 158
B.5 Summary ................. 159
List of Figures

1.1a Single stage compressor characteristics with different clearances (McDougall [7])
1.1b Multi-stage compressor characteristics with different clearances (Wisler [4])
1.2 Schematic of compressor rotor with tip clearance flow
1.3 Loss in peak pressure rise capability with tip clearance size (Smith [2])
1.4 Efficiency loss with tip clearance size (Wisler [5])
2.1 Schematic of the rotor-stator interaction in the multi-stage environment
2.2 Approximations to the rotor-stator interaction problem
2.3 Baseline rotor-bowed stator geometry
2.4 Conventional radial (straight) stator and three-dimensional bowed stator
2.5 Bowed and straight stator loss buckets measured in a multi-stage test compressor (Weingold et al [55])
2.6 Views of the complete computational domain
2.7 Comparison of compressor test data with NASTAR predictions for an 11-stage HPC (Lejambre et al [50])
2.8 Stage/rotor inlet profiles utilized for present investigation
3.1 Rotor pressure rise characteristics
3.2 Rotor mass averaged relative total pressure loss buckets
3.3 Rotor inlet axial velocity profiles
3.4 Rotor pitch averaged spanwise loss profiles
3.5 Axial planes at 2/3 rotor chord and tangential planes at the center of the rotor tip clearance gap
3.6 Rotor pitch averaged spanwise loss profiles at high and low loading
3.7 Rotor entropy rise with increased blade loading
3.8 Rotor flow blockage variation (data from Khalsa [70])
3.9 Stator pressure rise characteristics
3.10 Stator inlet axial velocity profiles at peak pressure rise
3.11 Stator mass averaged absolute total pressure loss bucket
3.12 Stator midspan diffusion factor variation with increased blade loading
3.13 Stage pressure rise characteristics
3.14 Comparison of the peak effective static pressure rise coefficient with the stalling pressure rise data of Koch [3]
3.15 Stage mass averaged adiabatic efficiency variation

4.1 Circumferential pressure variation imposed on the rotor by the downstream stator at one instant of time
4.2a Radial pressure variation imposed on the rotor by the downstream stator at one instant of time
4.2b Pitch averaged radial pressure profile imposed on the rotor by the downstream stator at one instant of time
4.3 Entropy contours at midspan at 4 instants of time showing periodic interaction of the rotor wake with the downstream stator
4.4 Rotor relative total pressure contours showing motion of the tip vortex in an axial plane mid-way between the rotor and stator
4.5 Rotor relative total pressure contours time averaged over 1 blade passing period
4.6 Rotor relative total pressure contours time averaged over 2 blade passing periods
4.7 Rotor relative total pressure contours obtained from steady flow averaging plane simulation
4.8 Stator leading edge static pressure time trace and corresponding FFT
4.9 Correlation of rotor tip vortex periodicity with rotor flow through time
4.10 Rotor pressure rise characteristic
4.11 Rotor spanwise loading distribution at peak pressure rise
4.12 Rotor back pressure profile obtained from the time averaged results and specified for the axisymmetric B.C. simulation at high loading
4.13 Rotor mass averaged relative total pressure loss bucket
4.14 Rotor pitch averaged spanwise loss profiles at high loading
4.15 Rotor mass averaged entropy rise variation
4.16 Maximum, time averaged, and minimum mass averaged entropy rise for the rotor at high and low loading

4.17 Maximum, time averaged, and minimum mass averaged entropy rise for the rotor tip region at high and low loading

4.18 Rotor tip clearance mass flow variation

4.19 Rotor tip clearance loss computed using the control volume model

4.20 Rotor tip region flow blockage correlation (data from Khalsa [70])

4.21 Rotor tip region flow blockage variation

5.1 Correlation of rotor tip vortex periodicity with rotor flow through time

5.2 Rotor pressure rise characteristic

5.3 Rotor mass averaged relative total pressure loss bucket

5.4 Maximum, time averaged, and minimum mass averaged entropy rise for the rotor

5.5 Bowed stator and corresponding straight (re-stacked) stator

5.6 Rotor pitch averaged spanwise loss profiles with bowed and straight downstream stators

5.7 Rotor spanwise loading distributions with bowed and straight downstream stators

5.8 Correlation of time averaged entropy rise for the rotor tip region with rotor tip loading

5.9 Maximum, time averaged, and minimum mass averaged entropy rise for the rotor tip region

5.10 Rotor tip region flow blockage correlation (data from Khalsa [70])

5.11 Rotor pitch averaged spanwise loss profiles with the downstream straight stator at nominal axial spacing and at 50% of the nominal axial spacing

5.12 Rotor pitch averaged spanwise loss profiles with the downstream bowed stator having (1r:1s) and (2r:3s) blade count ratios

5.13 Rotor pitch averaged spanwise loss profiles with the circumferentially specified downstream pressure variation

5.14 Time accurate response of an isolated rotor to a back pressure pulse

5.15 Pressure ratio as a function of time for the isolated rotor subjected to a back pressure pulse

5.16 Rotor pitch averaged spanwise loss profiles from the time accurate and steady axisymmetric B.C. simulations of the isolated rotor
B.1 Entropy contours at 97% span at 4 instants of time showing interaction of tip clearance vortex with the downstream stator

B.2 Stator pressure rise characteristic

B.3 Stator instantaneous spanwise loading distribution at peak pressure rise

B.4 Stator inlet flow angle variation

B.5 Stator mass averaged total pressure loss bucket

B.6 Stator phase corrected mass averaged entropy rise variation

B.7 Maximum, time averaged, and minimum phase corrected mass averaged entropy rise
List of Tables

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>Rotor design parameters</td>
<td>33</td>
</tr>
<tr>
<td>2.2</td>
<td>Stator design parameters</td>
<td>34</td>
</tr>
<tr>
<td>4.1</td>
<td>Rotor exit flow field parameters</td>
<td>75</td>
</tr>
<tr>
<td>4.2</td>
<td>Rotor performance parameters at high loading</td>
<td>87</td>
</tr>
<tr>
<td>4.3</td>
<td>Rotor performance parameters at low loading</td>
<td>87</td>
</tr>
<tr>
<td>4.4</td>
<td>Predicted rotor tip loss at high loading</td>
<td>93</td>
</tr>
<tr>
<td>4.5</td>
<td>Rotor efficiency ratios</td>
<td>94</td>
</tr>
<tr>
<td>5.1</td>
<td>Rotor exit flow field parameters</td>
<td>115</td>
</tr>
<tr>
<td>5.2</td>
<td>Time averaged rotor performance metrics</td>
<td>118</td>
</tr>
<tr>
<td>5.3</td>
<td>Fluctuation in rotor performance metrics</td>
<td>119</td>
</tr>
<tr>
<td>5.4</td>
<td>Rotor and straight stator parameters at 50% axial spacing</td>
<td>122</td>
</tr>
<tr>
<td>A.1</td>
<td>Steady flow NiSTAR simulations</td>
<td>152</td>
</tr>
<tr>
<td></td>
<td>Steady flow NASTAR simulations</td>
<td>153</td>
</tr>
<tr>
<td>A.2</td>
<td>Unsteady flow NiSTAR simulations</td>
<td>153</td>
</tr>
</tbody>
</table>
Nomenclature

\( a \) \hspace{1cm} \text{speed of sound} \\
\( A \) \hspace{1cm} \text{area} \\
\( A_n \) \hspace{1cm} \text{Fourier coefficient} \\
\( A_b \) \hspace{1cm} \text{blocked area} \\
\( b_z \) \hspace{1cm} \text{blade axial chord} \\
\( c \) \hspace{1cm} \text{blade chord} \\
\( c_p \) \hspace{1cm} \text{specific heat at constant pressure} \\
\( DF \) \hspace{1cm} \text{diffusion factor} \\
\( h \) \hspace{1cm} \text{specific enthalpy} \\
\( M \) \hspace{1cm} \text{Mach number} \\
\( n \) \hspace{1cm} \text{spatial harmonic number} \\
\( \dot{m} \) \hspace{1cm} \text{mass flow} \\
\( P \) \hspace{1cm} \text{pressure} \\
\( PR \) \hspace{1cm} \text{pressure ratio} \\
\( Q \) \hspace{1cm} \text{dynamic pressure} \\
\( R \) \hspace{1cm} \text{ideal gas constant} \\
\( Re \) \hspace{1cm} \text{Reynolds number} \\
\( s \) \hspace{1cm} \text{blade pitch, or specific entropy} \\
\( t \) \hspace{1cm} \text{time} \\
\( T \) \hspace{1cm} \text{time, or temperature} \\
\( TR \) \hspace{1cm} \text{temperature ratio} \\
\( U \) \hspace{1cm} \text{rotor wheel speed} \\
\( u, v, w \) \hspace{1cm} \text{velocity components in (x, \( \theta, r \))} \\
\( V \) \hspace{1cm} \text{velocity} \\
\( x \) \hspace{1cm} \text{axial coordinate} \\
\( y \) \hspace{1cm} \text{Cartesian azimuthal coordinate}
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\alpha$</td>
<td>tangential flow angle</td>
</tr>
<tr>
<td>$\beta$</td>
<td>relative frame tangential flow angle</td>
</tr>
<tr>
<td>$\gamma$</td>
<td>ratio of specific heats</td>
</tr>
<tr>
<td>$\epsilon$</td>
<td>tip clearance non-dimensionalized by chord</td>
</tr>
<tr>
<td>$\eta$</td>
<td>efficiency</td>
</tr>
<tr>
<td>$\theta$</td>
<td>circumferential coordinate</td>
</tr>
<tr>
<td>$\xi$</td>
<td>meridional flow angle</td>
</tr>
<tr>
<td>$\rho$</td>
<td>density</td>
</tr>
<tr>
<td>$\tau$</td>
<td>tip clearance size</td>
</tr>
<tr>
<td>$\phi$</td>
<td>flow coefficient (axial velocity/rotor speed)</td>
</tr>
<tr>
<td>$\chi$</td>
<td>mass flow ratio</td>
</tr>
<tr>
<td>$\psi$</td>
<td>pressure rise coefficient</td>
</tr>
<tr>
<td>$\bar{\omega}$</td>
<td>total pressure loss coefficient</td>
</tr>
</tbody>
</table>

**Subscripts:**

- 1  at rotor inlet
- 2  at rotor exit
- 3  at stator exit
- $avg$  average condition
- $axi$  from axisymmetric B.C. simulation
- $c$  compressor
- $cor$  corrected condition
- $CV$  control volume model
- $e$  at edge of defect region
- $exit$  at exit
- $f$  flow through value
- $i$  at station ($i$)
- $in$  at inlet
- $j$  tip clearance jet
- $m$  main flow
Profile region
radial component
isentropic state
relative frame stagnation condition
stagnation condition
tip region
total flow passage
from unsteady simulation
tip vortex region
axial component
Cartesian azimuthal component
circumferential component

Superscripts:

- averaged value
' perturbation
m mass average
t time average

Mathematical Operators:

i complex number $\sqrt{-1}$
$\Delta$ difference
$\partial$ partial derivative
$\nabla$ gradient
$\nabla^2$ Laplacian
$| |$ absolute value
$\| \|$ magnitude of a quantity
Abbreviations:

<table>
<thead>
<tr>
<th>Acronym</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>B.C.</td>
<td>Boundary Condition</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>CFL</td>
<td>Courant-Freidrichs-Lewy stability criterion</td>
</tr>
<tr>
<td>FFT</td>
<td>Fast Fourier Transform</td>
</tr>
<tr>
<td>HPC</td>
<td>High Pressure Compressor</td>
</tr>
<tr>
<td>NASTAR</td>
<td>Navier-Stokes for Arbitrary Regimes</td>
</tr>
<tr>
<td>NiSTAR</td>
<td>Ni’s Navier-Stokes for Arbitrary Regimes</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds-averaged Navier-Stokes Equations</td>
</tr>
</tbody>
</table>
1.1 Background and Motivation

It is well known that the performance and stability of an axial compressor depend strongly on the flow in the endwall region. This portion of the flow field is primarily associated with the rotor tip clearance, which is the gap between the rotating blade tips and the stationary outer casing. To illustrate the overall effects of tip clearance size, pressure rise versus mass flow characteristics are shown in Figures (1.1a) and (1.1b) for a single stage and a multi-stage compressor respectively. As the mass flow rate through a compressor is decreased the pressure rise across the machine increases until a performance limiting aerodynamic instability (compressor "surge" or "stall") is encountered [1]. When the tip clearance size is increased, the flow range over which the machine may operate free of these instabilities reduces as indicated in Figures (1.1a) and (1.1b). In addition, for a given flow condition, enlarging the tip clearance decreases the compressor pressure rise capability and the efficiency [2, 5]. Therefore, the design and off-design performance of the machine are affected by the rotor tip clearance flow.

Figure (1.2) shows a schematic of the tip clearance flow field. Due to the pressure difference across the rotor blade tip, leakage flow moves from the pressure side of the airfoil over the blade to the suction side. The clearance flow then rolls up to form a vortex somewhat analogous to the wing tip vortices on aircraft. The fluid in the tip leakage vortex is characterized by high stagnation pressure loss and low axial momentum. Hence it cannot
withstand a strong adverse pressure gradient without large increases in cross-sectional area, i.e. in aerodynamic “blockage”.

One of the earliest compilations of experimental data which demonstrated the importance of rotor tip clearance in determining multi-stage compressor pressure rise capability was that of Smith [2]. Figure (1.3) from [2] shows that, for a number of compressors with clearances between 1.5 and 8% of chord, a 1% increase in clearance/chord produces approximately a 5% decrease in peak pressure rise\(^1\). The trend shown has been corroborated by more recent data from other sources including Koch [3], Wisler [4, 5] and [6-9].

Not only pressure rise capability, but also efficiency is impacted by clearance as shown in Figure (1.4) from Wisler [5]. Efficiency penalties range from 1 to 2 points for each 1% increase in the stage average clearance/blade height, depending upon the design. Over the years, data such as these have provided strong motivation to understand the nature of the tip clearance flow field as well as develop techniques to modify it.

In spite of the importance of the tip clearance flow, however, complete explanations of the mechanisms by which the endwall flow degrades blade row performance still do not exist, and the present status is that empirical correlations are employed in the design process. In addition, the relevant fluid dynamic mechanisms are still poorly understood with regard to multi-stage compressors. Therefore, elucidating the aerodynamic mechanisms responsible for the impact of tip clearance flow may help in the development of multi-stage compressors which have improved performance.

1.2 Review of Previous Work

This thesis examines the unsteady flow within a compressor stage, in particular the impact of the downstream stator pressure field on the upstream rotor tip clearance flow.

\(^1\) For clearance/chord less than 1.5%, sensitivity to clearance change is also observed, however data appears less conclusive regarding the magnitude of the effect on pressure rise capability.
The topics chosen for review thus include tip clearance flows and blade passage scale unsteady flows. The review of previous work was limited to that which has had a strong influence on the present investigation.

1.2.1 Tip Clearance Flows

Traditionally, to gain insight into the flow processes occurring in the endwall region of multi-stage machines required detailed experimental measurements of the compressor flow environment.

With regard to experiments the work of Smith [2], Koch [3], Wisler [4, 5] and others have clearly demonstrated overall relationships between various geometric parameters and aerodynamic "figures of merit". As a result of such studies, one parameter which has emerged as controlling the behavior of a number of compressors is rotor tip clearance size. Examples of empirical correlations based on this are shown in Figures (1.3) and (1.4).

Other studies such as Smith and Cumpsty [6], Goto [8], Smith [10], McDougall [11], and Inoue et al [12] have focused on how the tip clearance flow impacts the blade passage aerodynamics through detailed measurements in the rotor endwall region. These investigations have provided connections between clearance size, rotor exit flow profiles, and overall blade row loss. Unfortunately, the behavior of the rotor endwall flow as influenced by different design variables is often difficult to discern from such data. In addition, quantitative measures of flow parameters such as tip clearance mass flow, rotor blade tip loading, and endwall loss and blockage are frequently unattainable. Because of this computational fluid dynamics (CFD), has been increasingly utilized to gain insight into the flow processes occurring within the compressor blade passages.

Most of the CFD investigations on endwall flow have focused on the steady flow behavior of isolated rotors or blade rows.
Early CFD investigations of tip clearance flow in high speed fans were conducted by Dawes [13] and Hah [14] in the mid-1980's. Studies like these served as proof-of-concept examples for the use of CFD to numerically compute endwall flows, but left much unresolved with regard to the fluid dynamic behavior of this portion of the compressor flow field.

The vortical structure of the tip clearance has been hypothesized to be an important feature and computational studies which focused on determining whether this was so include those of Chen et al [15] and Brookfield [16]. Using slender body theory, Chen et al [15] modeled the tip clearance flow as a two-dimensional unsteady flow. Similarity considerations lead to a trajectory for the tip clearance vortex which was shown to apply to a range of data. Using three-dimensional, steady, Reynolds-averaged Navier-Stokes simulations, Brookfield [16] showed that there were competing influences on the trajectory obtained by Chen, due to turbulent diffusion and streamline divergence. Brookfield [16] and later Khan [17] also examined the behavior of clearance type vortices in an adverse pressure gradient and showed that vortex breakdown was not a factor for representative compressor tip vortices.

Using both CFD and control volume analysis, the generation of loss in the endwall region was addressed by Storer and Cumpsty [18]. They found that most of the loss in the endwall region could be attributed to the mixing of the leakage flow with the main flow. Based on this, a control volume analysis was developed to compute the losses formed by mixing the leakage jet and the surrounding blade passage flow. The results were shown to agree with experimental data, indicating that the overall clearance related loss may not be strongly dependent upon the details of the endwall flow.

The computational studies of Khalid [19] and Adamczyk et al [20] focused on endwall blockage formation and its subsequent effects on compressor pressure rise capability. Khalid [19] established a quantitative connection between tip clearance flow and the formation of endwall blockage. This included a methodology to compute blockage in
the three-dimensional flow environment. Adamczyk et al [20] examined how endwall blockage forms in a high speed fan due to the interaction of the clearance vortex and in-passage shock. Based on this, it was concluded that flow over the forward portion of the rotor tip clearance gap controlled the processes leading to fan stall.

More recently, CFD analysis of flow in the endwall region of multi-stage compressors has been conducted by Graf and Sharma [21] and Dring et al [22]. For a single stage of a modern high pressure compressor, Graf and Sharma [21] examined the how radial pressure field due to a downstream stator affects the upstream rotor pressure rise capability and loss. For a given overall pressure rise and mass flow condition, differences of up to 10% percent in rotor relative total pressure loss were obtained by changing rotor back pressure profiles. This study demonstrated the importance of modeling adjacent airfoil rows for multi-stage simulations of tip clearance flow. Dring et al [22] assessed the influence of tip clearance size on the predicted pressure rise of a multi-stage compressor. Their results also indicated that simulation of neighboring blade rows has an effect on the radial distribution of loss predicted within a multi-stage compressor.

In addition to the computational studies of the endwall flow region, it should be mentioned that a number of analytical models have been formulated since the pioneering work of Rains [23]. The majority of these models may be cast into one of two categories: leakage flow models, or lifting line techniques. With regard to flow field prediction, the use of modern CFD has superseded many of the older methods, however, those models which clearly highlight the essential fluid dynamic mechanisms are still of value. A review of some of these approaches has been provided by Chen [24].

Finally, it should be emphasized that in all of the models and computational studies described above the rotor endwall flow has been approximated as steady or quasi-steady. Although this may be appropriate for isolated blade rows, and may be approximately true for certain embedded blade rows, a goal of the present study is to determine the effects of blade passage scale unsteadiness on the tip clearance flow in multi-stage compressors.
1.2.2 Unsteady Flows: Rotor-Stator Interaction

Flows in multi-stage turbomachines are inherently unsteady and there is now heightened interest in determining how unsteadiness impacts aerodynamic performance and durability of turbomachines. In part, work on this topic has been sparked by continuing demands to improve upon existing designs and design methodologies, and a recognition that advanced designs are in a region which tends to increase the impact of flow unsteadiness. The view is that an improved understanding of unsteady flows will eventually lead to increases in machine efficiency and stability margins, extended engine life-times, and lower component development times and costs.

According to a 1992 NASA workshop on unsteady flow [25], a research area which has the potential to lead to such improvements is the study of blade passage scale unsteady flows. For multi-stage axial compressors, a short list of unsteady flow phenomena on this scale may include:

- interaction of incident wakes with moving/stationary blades
- interaction of vortices (e.g. tip vortices) with moving/stationary blades
- pressure field interaction of adjacent blade rows.

Over the years numerous investigations have been conducted to assess various aspects of such phenomena and a brief review of work which has influenced the present study will now be given.

Some of the earliest theoretical investigations of blade passage scale unsteady flows include the studies of Kemp and Sears [26-28], which were based on the thin airfoil theory of Sears and Von Kármán [29, 30]. These were two-dimensional, inviscid, linearized and highly simplified, however some insight was gained regarding blade circulation and lift variations, shed vorticity, and the modulation of pressure forces in unsteady flow. A result shown in [26-28] was that the force and moment changes due to blade-wake interaction and blade row pressure field interaction were of the same order of magnitude. The analysis
was linearized and no conclusions were given regarding the importance of unsteadiness on the time averaged performance of blade rows.

Theoretical investigations which approximated blade rows using either lifting lines or arrays of moving vortices/vortex sheets include those of Preston [31], Hawthorne [32], Horlock [33] and Henderson and Daneshyar [34]. In these studies it was shown that for an unsteady flow passed a fixed (or moving) point in space, a change in the time averaged total pressure field could be brought about. These models were some of the first to indicate that unsteadiness could potentially impact the time averaged blade row performance. However, the focus of these investigation was primarily on determining the role of unsteadiness as it related to airfoil pressure field response, and little was said regarding changes in machine efficiency or loss.

Recent investigations which examine how inviscid blade row pressure field interaction changes the time averaged total temperature and total pressure within compressors include those of Shang et al [35] and Paulon et al [36]. The effects found were small and were linked to phase differences between the measured total pressures and total temperatures which occur in the unsteady flow environment. It was concluded that co-location of temperature and pressure sensors was required to accurately (within ~0.25%) determine blade row efficiency.

A key reference in the area of unsteady flow and its impact on blade row performance is that of Kerrebrock and Mikolajczak [37], who identified the phenomenon of total temperature segregation in compressor stages resulting from the migration of rotor wake fluid towards the pressure surface of the downstream stator. This unsteady flow effect leads to circumferential temperature profiles downstream of stators, and implies that to compute efficiency it is necessary to measure the pitchwise temperature distribution, and compare the average enthalpy rise to the pressure rise. These conclusions were subsequently verified by inert-gas wake tracing experiments of Kumar and Kerrebrock.
[38], as well as other experimental investigations including Stauter et al [39] and Zierke and Okiishi [40].

In general, the impact of individual sources of unsteadiness (i.e. wakes, vortices, and potential interactions) do not seem entirely separable. For example, it is not clear what unsteady fluid dynamic phenomena are present in the determination of optimal inter-blade row axial gaps in multi-stage compressors. In 1970, Smith [2] reported an efficiency increase of approximately 1% for reduced gaps in a four-stage compressor. Later, Mikolajczak [41] confirmed this finding, while experiments by Hetherington and Moritz [42] contradicted it. These experiments suggest that selecting an aerodynamically optimal axial spacing may not be a simple task.

During the past decade, advances in computer technology have made possible the simulation of blade scale unsteady flows in turbomachines. In general, much of this work has addressed numerical algorithm development and proof-of-concept demonstrations. However, the application of advanced viscous and inviscid codes to the study of fluid dynamic phenomena associated with blade scale interactions has become more prevalent. Recent examples include the works of Valkov [43] and Gundy-Burlet et al [44], who investigated compressor wake-blade interaction in single and multi-stage machines respectively, Takahashi and Ni [45] who studied turbine hot-streak migration, and Dawes’s [46] examination of wake and tip vortex interaction in a transonic compressor stage.

A related topic has been the modeling of deterministic (i.e. a multiple of rotor shaft speed) unsteady flows for incorporation into multi-stage CFD codes, which has been addressed by Adamczyk [47, 48]. This approach utilizes ensemble, temporal, and spatial averaging of the equations of motion to obtain Reynolds stress-like terms which account for the deterministic unsteady phenomena in turbomachines (see Section (2.4.2) for further discussion of this method). Inclusion of such models into a CFD design code has met with some success as reported by Rhie et al [49] and Lejambre et al [50].
Due to the volume and breadth of research in the area of unsteady flows in turbomachines, review papers have emerged as a useful resource for an overall assessment of the field. Reviews which span significant portions of the subject include those of Mikolajczak [41], Hetherington and Moritz [42], and Greitzer et al [51]. In addition, useful subject specific reviews which cover the areas of transition and aeroelasticity-aeroacoustics have been given by Mayle [52] and Verdon [53], respectively.

1.3 Problem Statement and Research Objectives

This thesis examines the impact of unsteady rotor-stator interaction on tip clearance flow and assesses the importance of blade passage scale unsteady flows in determining compressor performance.

The objectives for the investigation are:

- Assess the significance of downstream pressure field non-uniformity on upstream rotor tip clearance flow, and overall aerodynamic performance.
- Determine how the rotor pressure rise capability, flow capacity, loss, and blockage depend on the levels and distributions of exit pressure non-uniformity.
- Evaluate the time averaged rotor pressure rise capability, flow capacity, loss, and blockage at different aerodynamic operating conditions.

The specific fluid dynamic questions to be addressed are:

- How do the rotor clearance flow and tip vortex respond to the unsteady (time-varying) pressure field imposed by the downstream blade row? Is the response significantly different than that obtained assuming steady flow in the relative frame?
- How are rotor and stator performance (primarily aerodynamic loss and pressure rise) affected by unsteady rotor-stator interaction?
- How are the quantitative and qualitative results influenced by aerodynamic operating condition and relevant geometric design parameters (such as blade row axial spacing and blade count ratio)?
- What fluid dynamic mechanisms are responsible for changes (if any) in blade row performance associated with the unsteady flow environment?
1.4 Contributions of Thesis

Unsteady flow computations have been conducted to provide first of a kind quantification of the effects of stator pressure field on upstream rotor performance. The behavior of the rotor tip vortex in a high pressure compressor stage has been examined and the dependence of endwall loss and blockage on the level of exit pressure non-uniformity has been established. The time averaged results showed increases in overall rotor loss and decreases in tip clearance loss compared to an isolated rotor. In addition, unsteady motions of the tip leakage vortex on the order of the blade pitch were observed at the rotor exit. This behavior enhanced mixing in the endwall region, causing the time averaged clearance related loss to be close to that computed assuming fully mixed out conditions. These results suggest that steady multi-stage flow predictions should reflect the time averaged effects on rotor endwall flow structure and loss.

An assessment of three different steady and quasi-steady approximations frequently utilized to analyze compressor flow behavior has also been made. It was found that the existing steady flow methods were generally unable to accurately compute the magnitudes of the time averaged overall rotor loss, tip region loss, and endwall blockage. However, in all cases examined the steady flow models gave overall rotor pressure rise and flow capacity values which were close to the time averages obtained from unsteady flow simulations.

1.5 Outline of Thesis

The thesis is organized in the following manner:

Chapter 2 presents the overall approach and methodology and provides descriptions of the computational fluid dynamic codes utilized. Chapter 3 covers the steady flow simulations and the quantitative assessment of component performance. In Chapter 4, the details of the unsteady rotor-stator simulation are given along with qualitative and quantitative comparisons with steady flow results. Observations from Chapter 4 are
utilized to formulate a list of specific fluid dynamic questions regarding the generality and sensitivity of the unsteady flow results to different operating conditions and geometric changes. To resolve these questions a sensitivity study was conducted and the results of this are presented in Chapter 5 along with conclusions based on all of the unsteady flow simulations. Chapter 6 provides the summary and conclusions, implications of this study, and recommendations for future work.
Figure (1.1a): Single stage compressor characteristics (McDougall [7])

Figure (1.1b): Multi-stage compressor characteristics (Wisler [4])
Figure (1.2): Schematic of compressor rotor with tip clearance flow (casing not shown)

Figure (1.3): Loss in peak pressure rise capability with tip clearance size (Smith [2])
Figure (1.4): Efficiency loss with tip clearance size (Wisler [5])
2.1 Introduction

With the availability of modern computational fluid dynamics (CFD) codes, it has become possible to analyze and visualize the flow field in the compressor endwall region in some detail. Even through the use of advanced CFD, simplifying assumptions and approximations must still be made to obtain solutions for multi-stage machines. The most common of these is that the flow within the rotor passage (i.e. relative frame) is steady [18-22], but the degree to which this is warranted is now an item of increased interest. A problem which examines one specific aspect of this unsteady blade row interaction is to determine the effect of the downstream stator pressure field on the upstream rotor performance.

In this Chapter, the approach taken to address this problem is described. Brief discussion of the computational models used to simulate the flow fields is presented along with blade passage geometries and flow conditions. Finally, some issues relating to the computational fluid dynamic codes are addressed.

2.2 Approach: Computational Fluid Dynamics

The present investigation utilizes advanced computational fluid dynamic (CFD) models to define the influence of the downstream stator pressure field on the rotor flow features. In this context, CFD was primarily employed to deduce changes in flow field response resulting from blade row interaction, and the absolute levels of the various flow
quantities were of less interest than changes which occurred when the flow conditions were modified. A detailed calibration of the codes was therefore not critical as long as the computed fluid dynamic parameters of interest were representative of the published experimental data for multi-stage compressors. In addition, two separate CFD codes were utilized to provide predictions of the flow field which could then be compared. Doing this demonstrated that the fluid dynamic features of interest (i.e. the tip vortex and wake) were essentially independent of the code.

The approach adopted was to examine a single stage (rotor followed by a stator) of a parametrically representative high pressure compressor. Figure (2.1) provides an idealized schematic of the problem being addressed. To determine how the downstream stator pressure field influences the rotor, simulations of both the rotor and the stage flow fields were performed.

The pressure field in front of a typical stator row contains both radial and circumferential variations. The stator pressure field is also unsteady due to the varying inflow. Four different approximations were thus utilized to assess the role of unsteadiness in determining rotor flow field behavior:

(A) rotor alone with circumferentially uniform, radially varying back-pressure profile (i.e. axisymmetric case)

(B) rotor-stator interaction modeled with steady averaging plane technique (i.e. "industry" standard technique)

(C) rotor-stator interaction modeled with steady deterministic stress and bodyforce boundary conditions

(D) full unsteady interaction of the rotor and stator

Figure (2.2) shows schematics of the four configurations. By comparing results based on these four scenarios, the effects of unsteadiness and the adequacy of the different flow field models can be ascertained. The complexity of the approximation grows from axisymmetric modeling of the stator pressure field (A), to an averaging plane technique (B), to a sophisticated model which incorporates both the stator pressure field and any
corresponding flow blockage effects (C), and then to the full unsteady situation (D). Each of these cases will be considered at length in Chapters 3 and 4. The physical and computational modeling aspects of approximations (B) and (C) are given in Section (2.4) along with related issues.

This problem demands the use of state-of-the-art CFD codes and extensive computational resources. Because of mutual interest in the results, the Pratt & Whitney Division of United Technologies Corporation agreed to the use of their codes and computational facilities for the study.

### 2.3 Rotor and Stator Design Parameters

The high pressure compressor (HPC) stage selected for this investigation was to be representative of a stage from a current aero-engine design. The range of design variables was as follows:

- realistic stage design point loading; $\Delta P/Q = 0.4$ to 0.5
- subsonic blade tip Mach number; $M_{\text{Tp}} = 0.5$ to 0.7
- representative Reynolds number; $Re = 0.5 \times 10^6$ to $1.5 \times 10^6$
- conventional endwall configuration (no casing treatment)
- nominal tip clearance; $\tau = 1$ to 2% of chord
- realistic geometric configuration (i.e. solidity, stagger, axial spacing, etc.)

Based on these criteria, the stage selected for the study was the 13th stage (which is the 9th stage of the HPC) of the Pratt & Whitney PW4000 compressor, Build #10. Although largely conventional, this stage incorporates two innovative design concepts. The baseline geometry utilized for the majority of computations consists of a hub contoured rotor and a three-dimensional "bowed" stator. Figures (2.3a) and (2.3b) show this geometry.

---

2 Computations of the rotor alone and with a re-stacked straight stator are discussed in Chapter 5.
The rotor hub has an endwall contour which increases the flow at the root and promotes attached flow at off-design conditions [50]. Similarly, the bowed stator was designed to reduce suction surface-endwall corner separation through the development of radial flows within the stator passage which increase flow into this corner [55]. A conventional straight stator and the corresponding bowed stator are shown in Figure (2.4). The benefits of the bowed design compared with a conventional stator can be seen in the stator span-averaged loss variation data in Figure (2.5). The data were obtained from experiments conducted using a three-stage compressor, with stator loss deduced from measurements made across the third stator row [55]. The improvement in blade loss at both the operating line and near stall conditions demonstrates the overall effect of bowing.

The impact of both bowed stators and endwall contoured rotors on compressor performance has been discussed by Lejambre et al [50], Weingold et al [55] and Coons [56]. During the present investigation, these design features were utilized to ensure that over the range of interest, the flow in the rotor hub region and in the stator passage remained attached. This is important because it allowed the interaction of the rotor tip leakage flow with the adjacent stator pressure field to be isolated. Tables [2.1] and [2.2] list some of the design parameters for the baseline stage.

Table [2.1]: Rotor Design Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hub/tip ratio</td>
<td>0.88</td>
</tr>
<tr>
<td>Span/chord</td>
<td>1.47</td>
</tr>
<tr>
<td>Clearance/chord</td>
<td>0.016</td>
</tr>
<tr>
<td>Axial gap/chord</td>
<td>0.32</td>
</tr>
<tr>
<td>Maximum thickness/chord</td>
<td>0.069</td>
</tr>
<tr>
<td>Mean stagger angle</td>
<td>57.5°</td>
</tr>
<tr>
<td>Solidity</td>
<td>1.30</td>
</tr>
<tr>
<td>Blade tip Mach no.</td>
<td>0.54</td>
</tr>
<tr>
<td>Reynolds no. based on chord</td>
<td>1,070,000</td>
</tr>
<tr>
<td>Inlet duct length/span</td>
<td>1.0</td>
</tr>
</tbody>
</table>
Table [2.2]: Stator Design Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Span/chord</td>
<td>1.38</td>
</tr>
<tr>
<td>Maximum thickness/chord</td>
<td>0.076</td>
</tr>
<tr>
<td>Mean stagger angle</td>
<td>27.7°</td>
</tr>
<tr>
<td>Solidity</td>
<td>1.40</td>
</tr>
<tr>
<td>Exit duct length/span</td>
<td>1.0</td>
</tr>
<tr>
<td>Stacking</td>
<td>30 degree lean from radial in inner and outer 40% of span</td>
</tr>
</tbody>
</table>

In the computations the inlet and exit ducts were extended so the phenomena associated with unsteady interactions would have sufficient distance to decay. This helped ensure that the results were not influenced by the inflow and outflow computational boundaries. Views of the complete computational domain from the top (axial-tangential projection) and the side (axial-radial projection) are shown in Figure (2.6).

**Rotor-Stator Blade Count Ratio**

The actual rotor-stator blade count ratio for this stage is 92:116 (or 1:1.26). For the unsteady CFD simulations, this ratio is impractical from the perspective of computational resources, and as a first step a 1:1 blade count ratio was utilized for the baseline investigation. This blade count ratio is not representative of the majority of HPC stages, so later simulations then assessed the effect of blade count ratio using unequal (2:3) blade counts (see Chapter 5).

Because this study is focused on determining the impact of the stator pressure field on the rotor, the approach taken to obtaining a 1:1 blade ratio was to geometrically scale the stator. The stator scaling was accomplished by holding solidity and airfoil thickness-to-chord and changing blade aspect ratio, which resulted in blades with longer chords and lower aspect ratio.
To determine the magnitude of this geometric change on the stator pressure field, initial steady flow simulations of the scaled and original blade were conducted using an Euler code. This provided the information necessary to determine if the design point pressure distributions had been altered. Results confirmed that scaled stator pressure distribution and its upstream influence were similar (within 2% of one another) to the original. Thus, for the baseline study, stator scaling was appropriate.³

2.4 Description of Computational Fluid Dynamics Codes

The codes utilized in the present study are the three-dimensional Reynolds-averaged Navier-Stokes (RANS) solvers developed by Ni [58] and Rhie et al [49]. All computations conducted during this investigation were run on the massively parallel workstation network at Pratt & Whitney.

2.4.1 NiSTAR

The CFD code employed for the majority of the steady, and all of the unsteady flow simulations is the NiSTAR solver of Ni [58]. This RANS code is largely based on an earlier Euler solver [59], and utilizes an explicit time-marching scheme. The basis for the spatial and temporal discretization is the Ni-Lax-Wendroff method [60] implemented in a cylindrical coordinate system. This technique approximates the inviscid terms in the equations of motion using a cell vertex centered finite volume formulation. The method is second-order accurate in both space and time. For the viscous terms, the shear stresses are computed at the center of each control volume. A secondary control volume is then employed to determine the corresponding rates of change of the stresses. This provides second-order accuracy in space and first-order accuracy in time.

The viscosity and thermal conductivity coefficients in the Reynolds-averaged equations are the sum of the laminar and turbulent values. The turbulent viscosity is

³ The stator parameters in Table [2.2] are for the scaled 1:1 blade count ratio.
obtained from a standard implementation of the Baldwin-Lomax mixing length turbulence model for wall bounded flows [61]. Numerical stability is maintained in inviscid regions of the flow using a blend of second- and fourth-order operators (artificial viscosity) similar to that of Jameson et al [62].

**Steady flow Simulation with NiSTAR**

When utilized for steady flow computations, NiSTAR makes use of an explicit multi-grid method with local time stepping based on the CFL number. The solver is a true multi-block code which can compute the flow in multiple blade rows using averaging plane inter-row boundary conditions. The averaging technique is stream thrust based, and is similar to mixing methods employed by others (e.g. Dawes [63]). Upstream and downstream boundaries which do not connect adjacent blade rows (i.e. for a single stage the rotor inlet and stator exit), allow a choice of either standard axisymmetric boundary conditions or non-reflective boundary conditions similar to those of Saxer [64]. Back pressure downstream of the last row is specified to set the aerodynamic operating condition for the computation. When simulating the flow in a single blade passage, periodic flow boundaries are utilized on both sides of the blade passage.

**Unsteady Flow Simulation with NiSTAR**

When utilized for unsteady flow computations, NiSTAR makes use of a time accurate technique with global time stepping. The time step size employed is based upon the flow through time associated with the smallest cell in the computational domain. The CFL condition on stability is relaxed through the use of implicit residual smoothing similar to that of Martinelli and Jameson [65]. Open boundaries upstream and downstream, and passage periodic boundaries are represented in the same manner as those for steady flow simulation. For these simulation inter-blade row information is passed without averaging.
permitting flow structures and pressure field effects to be felt by the neighboring airfoil rows.

*NiSTAR Computational Grid*

For all the baseline steady and unsteady flow simulations, a single computational grid was utilized.\(^4\) This ensured that comparisons made between cases to identify changes in the flow field would be independent of the mesh. However, a study was conducted to assess the impact of grid resolution on various flow parameters, as related in Chapter 3.

The standard grid employed was a structured H-mesh with dimensions 113x33x73 for the rotor and 113x33x73 for the stator. For the rotor, flow in the tip clearance gap was solved using an embedded grid block which places the clearance mesh within the base rotor grid. The tip grid was 46x8x8, hence 8 of the 73 radial planes in the rotor were located within the clearance region. The axial distribution of grid points for the rotor had 45 upstream, 47 in the passage, and 21 downstream. Similarly, for the stator there were 21 points upstream, 47 in the passage, and 45 downstream. When simulating steady flow through the stage this resulted in 3 levels of multi-grid being employed.

The topology of the mesh was such that at the rotor-stator interface the grid points were equally spaced in both the radial and tangential directions. For unsteady flow computations this reduced interpolation error at the sliding boundary and provided uniform spatial resolution of unsteady flow phenomena entering and leaving the interface.

*Computational Time and Convergence*

The simulations were conducted primarily on Sun Sparc model 51 and 61 workstations. A parallel computing environment was utilized with each blade passage being assigned to a separated workstation. Convergence of the code was determined by

\(^4\) The baseline case is the rotor-bowed stator (1:1) with extended inlet and exit ducts. Modifications of the original grid to examine other geometries are given in Chapter 5.
examining both numerical residuals and changes in fluid dynamic parameters of interest. Fluid dynamic parameters were monitored in hierarchical order beginning with those which converged most rapidly (e.g. mass flow and static pressure ratio) and continuing through to quantities which converged more slowly (e.g. total pressure and total temperature ratio, efficiency). Convergence of the steady code was declared when the individual blade row mass averaged total pressure and total temperature ratios changed by less than 0.1% despite continued computation. Generally this ensured that the changes in stage efficiency associated with numerical variations were an order of magnitude smaller ($\Delta \eta \sim 0.01$) than those of interest. Convergence of the unsteady code was based on obtaining periodic behavior of the aerodynamic parameters of interest; this is discussed further in Chapter 4. For the baseline stage configuration, typical convergence times for steady and unsteady simulations were 50 and 1050 CPU hours, respectively.

2.4.2 NASTAR

The second multi-stage RANS code which was employed only for steady flow computations of the baseline rotor-stator configuration, is the NASTAR solver developed by Rhie et al [49]. NASTAR is a three-dimensional flow code specifically designed for turbomachinery applications. As such it employs bodyforce and deterministic stress modeling to simulate the multi-stage environment. In the present study, this code was utilized to provide solutions which could provide an independent assessment of the steady NiSTAR solutions.

The NASTAR code is a finite volume, implicit, pressure correction RANS solver. Numerical modeling is based on the discretization of the fully conservative form of the governing equations in generalized curvilinear coordinates. The formal discretization is second-order accurate in space using centered differencing. To model the Reynolds stress and Reynolds heat flux terms, the turbulent viscosity and turbulent thermal conductivity are introduced. These are determined using a two equation $\kappa-\epsilon$ turbulence model. The near
wall regions are modeled by the Van Dreist near wall formulation of Dash et al [66]. Numerical stability is maintained through the use of artificial dissipation. For the continuity equation, control volume face mass fluxes are constructed using pressure weighted interpolation [67]. For the other equations, locally varying second-order dissipation is employed to stabilize the solution without sacrificing numerical accuracy [68].

The solution procedure begins by solving the momentum equations with a preliminary pressure field. Since the corresponding preliminary velocity field does not satisfy the continuity equation, the pressure equations are solved to establish new velocity and pressure fields which satisfy continuity. The momentum and continuity equations are coupled through this pressure correction procedure. A three-step pressure correction method is utilized with the energy and turbulence scalar equations being solved in turn [67]. All of the equations are solved using the successive line under relaxation technique.

**Bodyforce and Deterministic Stress Boundary Condition Modeling**

As described by Rhie et al [49] the shortcomings of the mixing plane approach, particularly for situations of strong reverse flow, motivated the NASTAR developers to consider alternate methods for simulating the multi-stage environment. The procedure adopted to model these effects employed a combination of bodyforce modeling and the deterministic stress approach outlined by Adamczyk [47, 48].

Bodyforces were utilized to simulated the effect of the neighboring downstream blade row radial and axial pressure field on the upstream blade row. In the bodyforce method used by NASTAR the circumferentially averaged static pressure at the mid-axial gap location observed in the downstream blade row solution is specified as the target back pressure for the upstream row. Blade forces from downstream are imposed in an axisymmetric bodyforce region located behind the upstream row (see Figure (2.2)). These blade forces make the static pressure field at exit of the upstream row coincide with that immediately ahead of the downstream row. The (circumferentially uniform) bodyforces are
computed by projecting the suction and pressure surface static pressures of the downstream airfoil to the mean camber line and integrating. The tangential bodyforces produce work/turning in the downstream rotor/stator. Thus, in an axisymmetric manner, the pressure field ahead of the downstream row is felt by the upstream row.

To model the viscous effects associated with the interaction of neighboring blade rows (e.g. wakes, separations and vortices), a procedure based on the deterministic stress formulation of Adamczyk [47] was employed. Adamczyk [48] defines the class of unsteady flows whose characteristic frequency is an integral multiple of shaft rotational speed as deterministic. Thus, wake passing would be a deterministic phenomena, while turbulence and vortex shedding are non-deterministic. To obtain equations which capture the time averaged effects of deterministic phenomena Adamczyk [47] begins with the Navier-Stokes equations and through a series of ensemble, temporal and spatial averaging, derives Reynolds stress-tensor-like terms known as deterministic stresses. The final result of this procedure yields the average-passage equation set.

The NASTAR adjacent blade row modeling is based on this approach, although not all of the mathematical terms in Adamczyk's [47] full average-passage equations are included (see [49] for a description of the terms which have been included). In addition, the NASTAR model does not account for any "real" unsteady flow effects such as circumferential pressure field interaction and unsteady wake transport. The NASTAR model can thus be regarded as a quasi-steady approximation of the unsteady flow problem. For details of the deterministic stress formulation references [47] and [49] are recommended.

**Steady Flow Simulation with NASTAR**

For the simulations conducted, the flow in the rotor passage, tip clearance gap, and downstream stator were computed. The downstream stator axisymmetric bodyforces and deterministic stresses were placed in an extended grid region located behind the rotor (see
Figure (2.2)) and were automatically updated during the solution process. The computational procedure required that a target mass flow rate be specified for each blade row. Based on this, the back pressure level at the exit of each row was independently adjusted while fixing the radial profile shape. Through iteration the pressure level was modulated until the specified flow rate was obtained. When converged the pressure profiles between the rows were required to match in a continuous manner and were no longer modulating despite additional computation. As with NiSTAR, NASTAR employs standard blade passage periodic boundary conditions to represent adjacent airfoil through flow behavior.

**NASTAR Computational Grid**

All the NASTAR simulations were conducted using the baseline stage geometry and a single computational grid. The grid employed was a structured H-mesh with dimensions 172x36x62 for the rotor, and 143x36x62 for the stator. For each axisymmetric grid sheet, the interior of the airfoil was initially filled with grid. However, those points located within the blade profile were not utilized, while those in the tip gap were activated. The mesh in the tip clearance space was 62x5x21, with 21 of the radial planes in the base rotor grid located in this region. The axial distribution of grid points for the rotor had 55 upstream, 62 in the passage, 17 between the trailing edge and the stator bodyforce region, 28 in the bodyforce region, and 10 downstream of this. Similarly, for the stator there were 26 points upstream, 62 in the passage, and 55 in the downstream duct.

**Computational Time and Convergence**

The simulations were conducted using a parallel computing environment. Ten workstations were simultaneously utilized for each single stage computation. The rotor and stator rows were each broken into five equal size portions which were assigned to separate workstations. Similar to NiSTAR, fluid dynamic parameters were monitored to determine
convergence. Convergence of the steady code was declared when the individual blade row mass averaged total pressure and total temperature ratios change by less than 0.1%. Again this ensured that changes in stage efficiency associated with numerical variations were an order of magnitude smaller than those of interest ($\Delta \eta \sim 0.01$). For each point along the speedline, the two row HPC simulation converged in approximately 60 to 80 hours of computation.

2.5 Code Validation

The NASTAR solver is the primary CFD tool used in the compressor design system at Pratt & Whitney [50]. As such, there have been numerous studies to validate and calibrate the solver. Examples are the 11-stage HPC simulations of Lejambre et al [50] and Gleixner [57], which showed good agreement with multi-stage rig data. Figure (2.7) from Lejambre et al [50] shows a comparison of predicted and measured total pressure and temperature ratios at inlet to the 8th stage (out of 11). In terms of stage averaged pressure and temperature ratio, both the average level and the variation of predicted profiles are within 1% of those measured.

Inlet Flow Profiles

The aerodynamic behavior of a blade row is affected by both the inlet and exit conditions. The present work addresses the influence of exit conditions only, while keeping the inlet flow circumferentially uniform and time invariant. This has been done to allow the effects of exit unsteadiness to be ascertained without the added complexity of inlet variations. The inlet profiles required for all aerodynamic simulations of the stage include total pressure, total temperature, circumferential flow angle, and meridional flow angle. To obtain such information, a combination of experimental data and multi-stage computations was utilized. Gleixner [57] conducted simulations of the 11-stage PW4000 HPC using the NASTAR flow solver. The inlet and exit conditions were specified based on experimental
data. Comparisons of shape and level of total pressure and total temperature profiles at the 8th stage were shown in Figure (2.7). Although experimental data was unavailable at inlet to the 9th stage (which is the stage utilized in the present investigation), the profiles computed by the multi-stage simulation of Gleixner [57] were thought to be realistic and adequate based on the agreement in Figure (2.7).

The inlet profiles obtained from the multi-stage calculations had to be adjusted to account for the extended rotor inlet duct. This was done by performing an axisymmetric flow computation for the duct with the multi-stage profiles specified as the exit conditions. This allowed determination of the inlet profiles. The absolute frame total pressure, total temperature, circumferential flow angle (α), and meridional flow angle (ξ) profiles deduced from this procedure are shown in Figure (2.8). These were specified at the stage/rotor inlet for all of the simulations conducted in this study. The profiles are given in dimensional form to emphasize both the absolute levels of the flow parameters encountered in the multi-stage flow environment, and the radial variations.
Figure (2.1): Schematic of rotor-stator interaction in the multi-stage environment

(A) Steady with axisymmetric exit profile

(B) Steady with averaging plane approach

(C) Quasi-steady with axisymmetric bodyforces and deterministic stress models

(D) Full unsteady interaction

Figure (2.2): Approximations to the rotor-stator interaction problem
(a) Overall stage geometry (casing not shown)

(b) Rotor and stator geometry at midspan

Figure (2.3): Baseline rotor-bowed stator geometry
Figure (2.4): Conventional radial (straight) stator and three-dimensional bowed stator

Figure (2.5): Bowed and straight stator loss buckets measured in a multi-stage test compressor (Weingold et al [55])
(a) Top view at midspan: axial-tangential plane projection

(b) Side view: axial-radial plane projection

Figure (2.6): Views of the complete computational domain
(a) Total temperature profile at inlet to 8th stage

(b) Total pressure profile at inlet to 8th stage

Figure (2.7): Comparison of compressor test data with NASTAR predictions for an 11-stage HPC (Lejambre et al [50])
Figure (2.8): Stage/rotor inlet profiles utilized for present investigation
Chapter 3

Steady Flow Simulations

3.1 Introduction

This chapter presents qualitative and quantitative descriptions of the flow in the baseline compressor stage under the assumption of steady flow.\(^5\) These were obtained by examining the stage behavior along a constant corrected speed pressure rise characteristic (i.e. a "speedline") using the NiSTAR and NASTAR steady flow solvers. The use of these two separate codes to perform the steady flow computations allowed assessment of the averaging plane and deterministic stress/bodyforce models described in the previous Chapter. The results of the computations will also be utilized in subsequent comparisons with unsteady flow simulations which examine the ability of the present steady flow models to capture the time averaged rotor and stator performance.

In addition, because the NASTAR solutions have been previously compared with experimental data (see Section (2.5)), evaluation of the NiSTAR predictive capability can be made through a direct comparison of results from the two computations. Although agreement between the two codes is not crucial to obtaining the goals of this investigation, it lends further confidence to the NiSTAR results as being representative of the actual compressor flow environment. Finally, it is essential that both an adequate description and quantification of the steady flow within the stage be obtained since this will be contrasted with the unsteady flow simulations of Chapters 4 and 5. The steady flow results thus serve as the baseline for the study and as a validation of the NiSTAR code.

\(^5\) Baseline stage refers to the rotor-bowed stator with (1:1) blade count ratio.
3.2 Rotor Flow Field

The behavior of the flow within the rotor blade passage is the focus of this investigation. Results for the downstream stator will therefore only be presented in the context of their impact on the rotor aerodynamics.

3.2.1 Rotor Performance Characteristics

The computed rotor pressure rise characteristics for the baseline stage, using the NiSTAR and NASTAR codes, are given in Figure (3.1). The inlet conditions are those given in Section (2.5). For the NiSTAR speedlines, the upper limit on loading was found by carrying out computations at higher levels of stage exit pressure until flow reversal occurred at the inter-blade row averaging plane. This caused the code to diverge, since information in the reversed flow region could not be re-computed and passed from the downstream stator to the upstream rotor. For the NASTAR solver, the upper limit on loading was deduced by decreasing the target mass flow until the stage pressure and temperature ratios were found to drop. This was also associated with the formation of separation or reversed flow within the stage.

The speedlines in Figure (3.1) show that the NiSTAR solver with the baseline grid produces a flatter pressure rise characteristic, with lower maximum loading, than that obtained from NASTAR. This difference at the maximum loading condition is approximately 13% in static pressure rise coefficient ($\Delta P/(P_{rel, in} - P_{in})$) and 3% in flow coefficient ($V_x/U$). The NASTAR speedline extends further since the range of solutions is not limited to those with no flow reversal downstream of the rotor. The bodyforce/deterministic stress grid extension which is used to represent the downstream stator in NASTAR allows small regions of reverse flow to extend downstream. Thus, given rotors which have small regions of flow reversal, NASTAR computations will generally yield higher peak pressure rise predictions than NiSTAR.
The lower peak pressure rise predicted by NiSTAR indicates higher losses and reduced turning are present in the flow field. Figure (3.2) shows the mass averaged rotor relative total pressure loss as a function of inlet flow angle (i.e. the loss "bucket"), computed by taking the difference in relative total pressure between stations (1) and (2) shown in Figure (2.6b) and normalizing by the inlet dynamic pressure. The higher losses associated with NiSTAR are in correspondence with the reduced pressure rise capability and account for part of the difference in rotor pressure rise capability predicted by the two codes. The loss buckets show increases as both peak loading (positive incidence) and below design pressure rise (negative incidence) are approached. The flow field solutions at the lowest loading condition were found to contain rotor pressure surface separations which are responsible for the increase in loss observed at this end of the loss bucket. The rotor pressure rise and loss variation from both baseline NiSTAR and NASTAR simulations are within 15% of one another over the range examined and provide quantitatively similar trends.

To resolve the differences between the two codes, a more detailed examination of flow fields was conducted. Figure (3.3) shows the pitch averaged rotor inlet axial velocity profiles for points (A) and (B) on the speedlines. The difference in the velocity profiles does not account for the difference in pressure rise obtained from the NiSTAR and NASTAR solutions, suggesting that the flow behavior within the rotor may be responsible for the observed differences in loss and pressure rise.

As an initial observation, the baseline rotor pressure rise characteristics from the two codes appear similar to experimental data obtained for a compressor having two different levels of tip clearance (e.g. Fig (1.1)). The inference drawn is that the phenomena responsible for the differences between NiSTAR and NASTAR may be associated with the rotor tip clearance flow. To investigate this, the rotor pitch averaged spanwise loss distribution is shown in Figure (3.4) at the (A) and (B) operating points. The greatest

---

6 The definition of loss coefficient is discussed at length in Section (3.2.3).
difference in the predicted rotor loss distributions appears over the outer ~20% of the span, implying that the tip region flow is responsible for a portion of the difference in rotor loss and pressure rise capability. In contrast, the flow along the inner 80% of the span appears attached with a relatively low level of loss. As stated in Chapter 2, this condition was a prerequisite for the investigation since it provides a favorable environment in which to examine the interaction of the tip vortex with the adjacent stator pressure field. Figure (3.4) suggests that at peak pressure rise flow reversal occurs in the tip clearance flow region (i.e. low relative total pressure region) and not along the blade. Thus the tip region appears ultimately responsible for the maximum obtainable rotor loading.

3.2.2 Effect of Computational Grid Resolution

As shown in Figure (3.4), the favorable agreement between the NiSTAR and NASTAR loss profiles along the inner 80% span indicates that for attached flows, the turbulence models (Baldwin-Lomax in NiSTAR, κ-ε in NASTAR) had little impact on the overall performance predicted for the rotor. However, the differences in loss occurring in the rotor tip region indicated that the computational grid utilized for the baseline NiSTAR simulations might have been too coarse. A grid refinement study was thus conducted. Based on the initial simulations the original NiSTAR rotor grid was modified to improve flow field resolution in the blade tip gap and near casing regions.

The modified rotor grid with tip clearance had 113x41x73 (original grid: 115x33x73) where 8 additional planes have been added in the tangential direction. The axial distribution of grid points is the same as the original, however, the radial distribution was adjusted to place 24 of the 73 planes in the rotor tip gap region. The new base rotor mesh had 113x41x49 along the blade, while the new embedded tip block had 46x24x24 (original tip block: 46x8x8). The total rotor grid point count was therefore increased from 272,217 to 338,209 with the number of points in the tip gap increasing from 2,994 to
26,496. The modified dense NiSTAR grid was similar to the NASTAR grid (see Section (2.4.2)) and was the largest possible based on available computational resources.

NiSTAR computations for the stage with the dense rotor grid were conducted at two points along the coarse grid speedline, corresponding to the maximum loading condition (point A) and the next point down. The results from the dense grid simulations are also shown in Figures (3.1) to (3.4). As compared to the coarse grid, the speedline shows increased pressure rise and the loss bucket shows a reduction in overall rotor loss. The results are now within approximately 5% of the NASTAR predictions. The inlet axial velocity profile is also closer to the NASTAR prediction and the rotor spanwise loss distribution is now nearly the same as that obtained from NASTAR.

To highlight the similarity of the NiSTAR, NiSTAR dense grid, and NASTAR results, Figures (3.5a) and (3.5b) show selected flow variables in axial planes at (2/3) chord and constant radius sections at the center of the tip gap for the rotor at operating points (A), (B), and (C). The contours of relative total pressure in Figure (3.5a) demonstrate that the most visible difference with the coarse grid occurs in the tip region. The dense grid NiSTAR and NASTAR predictions yield similar loss distributions and flow structures. The coarse grid result appears qualitatively like a machine having a slightly larger tip clearance flow. Based on the loss in peak pressure rise capability from the dense to the coarse grid, the difference in effective tip clearance size would be approximately 1% in clearance/chord. The flow features along the remainder of the span (i.e. wake and hub region) are similar despite differences in the grids and codes. The changes in the NiSTAR grid thus primarily affect the clearance related flow field, with little impact outside the tip region.

The rotor static pressure at the center of the clearance gap is shown in Figure (3.5b) and indicates the trajectory of the tip leakage vortex. The location at which the leakage

---

7 The modified dense grid does not go further along the speedline than the baseline NiSTAR case because it is also limited by reversal of flow at the interface plane. In fact, due to the improved resolution the solution encounters flow reversal sooner, hence the speedline does not extend as far as the coarse grid case.
vortex originates (~1/4 chord) as well as the subsequent tangential migration appears alike in all three cases.

In summary, these results demonstrate the similarity of the NiSTAR and NASTAR predicted flow fields despite differences in the computational procedure and modeling. In addition, the coarse grid NiSTAR solution has been shown to give results which are qualitatively consistent with those for a machine having slightly larger tip clearance. The NiSTAR code with the baseline grid will therefore be utilized for all subsequent simulations and analyses since it provides realistic performance trends and flow structures at acceptable computational cost.

3.2.3 Aerodynamic Losses

Quantifying rotor loss in the steady flow environment is important because it provides a figure of merit which can be utilized to assess the effects of unsteady blade row interaction. The rotor loss bucket in Figure (3.2) provided the overall integrated loss variation with inlet flow condition, but did not illustrate how the radial distribution of loss was impacted as the stage was throttled. Figure (3.6) shows the pitch averaged spanwise distribution of loss coefficient at points (A) and (D) along the speedline. The rotor tip region (outer ~10% span) mass averaged loss has increased by approximately 40% while that along the remainder of the blade stays approximately the same. The slightly lower loss in the hub region (inner ~10% span) at increased loading was found to be linked to a decrease in size of a hub suction surface corner separation which occurs because the increase in rotor tip region blockage causes flow to shift radially toward the hub. Figure (3.6) shows that the rotor tip region loss exhibits the greatest sensitivity to blade loading.

Thus far the figure of merit used to quantify the aerodynamic performance has been the loss coefficient defined as the change in mass averaged relative total pressure normalized by the difference between the mass averaged relative total pressure and static pressure at inlet to the blade row (i.e. \( \Delta P_{rel}/(P_{rel,in} - P_{in}) \)). This coefficient is often
utilized in the industry because of its simplicity and its usage as a measure of cascade performance. However, rotor relative total pressure is strictly conserved only in two-dimensional, inviscid, steady flow. The flow fields of interest in the present investigation are three-dimensional, viscous, and in some cases unsteady. Thus a more appropriate measure of blade row aerodynamic loss must be utilized.

Fundamentally, the quantity of interest in determining the aerodynamic loss is entropy rise [69]. For flow through a blade row the difference in mass averaged entropy flux (from inlet to exit) can be written as,

$$\Delta \bar{s}_{rot} = \frac{1}{\dot{m}_{rot}} \int_{Total} (s_2 - s_1) \, d\dot{m}_{rot}$$  \hspace{1cm} (3.1)

This definition is appropriate even when applied to flows with radial and tangential non-uniformities, viscosity, and unsteadiness, however the value obtained is dependent upon the locations of stations (1) and (2). To overcome this dependence some studies utilize loss coefficients based on mixing the flow to a uniform state. In the present work, it is of interest to examine the loss across blade rows which are closely coupled. Because the rotor and stator are tightly spaced, with non-uniform flow occurring between them, a “mixed out” definition for loss coefficient over-estimates (by 20 to 30%) the loss within the rotor and stator passages. In addition, although a “mixed out” coefficient would not be affected by the location of stations (1) and (2), it is not readily apparent how such a mixing process should be defined in three-dimensions (e.g. constant pressure, radial equilibrium, constant area, etc.). Thus the application of equation (3.1) to the steady flow and on an instantaneous basis to the unsteady flow, is used here to compute the aerodynamic loss generated up to the plane of interest.

In all computations to be presented, the entropy rise through the rotor is computed using axial planes at stage inlet (station 1) and at the middle of the rotor-stator axial gap (station 2). For the stator, the entropy rise is computed from the middle of the rotor-stator
axial gap (station 2) to an axial plane located 10% chord downstream of the stator (station 3). These locations are illustrated in Figure (2.6).

As shown in Figure (3.6), the loss in the rotor tip region increases with loading changes. Quantification of the entropy rise in this portion of the flow field thus provides one measure of rotor sensitivity to the downstream stator pressure field. For the rotor the entropy rise will be broken into two parts,

\[
\Delta \bar{s}^m_{\text{Tot}} = \frac{1}{m_{\text{Tot}}} \left[ \int_{T_{\text{pp}}} (s_2 - s_1) \, d\dot{m}_{\text{Tip}} + \int_{\text{Profile}} (s_2 - s_1) \, d\dot{m}_{\text{Pro}} \right]
\]

(3.2)

\[
= \frac{1}{\bar{m}_{\text{Tot}}} \left[ \dot{m}_{\text{Tip}} \Delta \bar{s}^m_{\text{Tip}} + \dot{m}_{\text{Pro}} \Delta \bar{s}^m_{\text{Pro}} \right]
\]

(3.3)

where the tip region is defined using the method of Khalid [19] (see Section (3.2.4)) and the profile region is the remainder of the flow. The entropy at a given station (i) is defined as that for a perfect gas,

\[
s_i = c_p \ln(T_n) - R \ln(P_n)
\]

(3.4)

\[
= R \left[ \frac{\gamma}{\gamma - 1} \ln(T_n) - \ln(P_n) \right]
\]

where \( R \) and \( \gamma \) are the mass averaged values at location (i). The total pressure and temperature utilized in (3.4) are those for flow in the frame of reference relative to the blade being considered (i.e. rotor relative or stator absolute). Computationally, the integrals are performed using trapezoidal rule integration in an axially aligned \((r-\theta)\) plane at the station specified.

Applying equations (3.2) through (3.4) to rotor flow field at points along the baseline NiSTAR speedlines yields the entropy rise variation shown in Figure (3.7). The rotor entropy rise has been normalized by the maximum value found over the range of loading examined, so Figure (3.7) shows the fraction of entropy rise associated with the tip region and the blade midspan. The overall rotor entropy rise increases at both high and low loading, with trends similar to that for the total pressure loss coefficient in Figure (3.2). At

57
low loading, the tip region loss is reduced because the amount of leakage flow is dependent on the airfoil loading, and the clearance loss is known to scale with this [18, 69]. In contrast, the profile loss increases at low loading due to pressure surface separation and enlarged hub corner separations which result from negative incidence operation. As the blade loading is increased towards the design point \((\Delta P/(P_{rel, in} - P_{in}) = 0.36)\), the profile loss approaches a minimum. Further increases in loading produce both profile and tip loss increases, with the tip region increasing more rapidly than the profile region. The growth in profile loss is associated with thickened suction surface boundary layers, while the rise in tip loss indicates the response of the rotor clearance vortex to the average pressure rise imposed by the downstream stator [21]. Figure (3.7) demonstrates that for steady flow through this rotor, profile loss dominates the overall loss behavior at low loading, while tip losses become increasingly important with elevated loading. In addition, since increases in rotor loading cause the tip region loss to increase more rapidly than the profile loss, this suggests that the clearance flow behavior influences the rotor peak pressure rise capability.

3.2.4 Aerodynamic Flow Blockage

In addition to rotor loss another measure of blade row sensitivity to tip clearance is the aerodynamic blockage associated with this region of the flow field. As shown by Khalid [19], Adamczyk [20], and Khalsa [70], the growth of tip region blockage provides an indication of the onset of blade performance limiting instability. As discussed in [1], the present ideas of rotating stall link its occurrence to the peaking over of the pressure rise characteristic. For compressors in which the tip region is primarily responsible for blockage growth, the onset of instability is thus linked to phenomena associated with the clearance flow. Therefore, quantification of the rotor endwall blockage provides support to the hypothesis that it is the tip region flow which often determines the rotor peak pressure rise capability.
As defined by Khalid [19], the equivalent blocked area associated with the tip clearance flow is,

$$A_b = \int_{\Gamma_p} \left(1 - \frac{\rho V_m}{\rho_x V_c}\right) dA \cdot \cos(\beta_e) \quad (3.5)$$

where the tip region is that associated with the total pressure defect of the endwall flow, and the blocked area is computed in a plane orthogonal to the vortex trajectory. Similar to the traditional definition of displacement thickness, the velocities $V_m$ and $V_c$ are those in the defect region and at the edge of the defect respectively. The velocity $V_m$ is taken in the direction of the main flow and the defect region is defined by Khalid [19] to be that with

$$\left\| \nabla(\rho V_m) \right\|_{\rho_{avg} V_x / c} > 2 \quad (3.6)$$

and

$$\left\| \nabla(\rho V_m) \right\| = \sqrt{(\nabla_r [\rho V_m])^2 + (\nabla_0 [\rho V_m])^2} \quad (3.7)$$

In the present study, when tip blockage is computed the tip region is radially limited to that associated with elevated loss (see Figure (3.6)) and excludes the rotor wakes. These restrictions are eliminated when computing the total passage flow blockage.

The rotor overall blockage and tip blockage were computed at the rotor-stator mid-axial location for each loading condition along the baseline NiSTAR speedline to establish the behavior in the steady flow environment. Figure (3.8) gives these results along with experimentally measured tip blockage from the low speed single-stage compressor measurements of Khalsa [70]. These measurements provide an additional assessment of the computations. The blockage area, normalized by the "over tip" clearance area ($\tau c$), is plotted against the tip region loading parameter (defined by Khalsa [70] as the sum of the rotor endwall region pressure rise and total pressure loss mass averaged along the blade chord and normalized by the dynamic pressure 2 clearance heights from the endwall). This correlation was utilized because it has been shown to collapse the data [70], and allows comparison of results for machines with various levels of tip clearance [19].
In Figure (3.8) both the data and the computations indicate that as the tip loading parameter increases towards approximately 0.8, there is a rapid growth in tip blockage. In general the rotor tip blockage accounts for 30 to 60% of the total passage blockage. Near peak loading the computations show that the growth rate for the tip blockage is increasing more rapidly than that of the overall blade passage. As shown by Khalid [19] and Khalsa [70] the rate of increase of endwall blockage is a measure of the sensitivity of the machine to tip clearance. The agreement between the experimentally measured blockage and that obtained from NiSTAR lends further confidence to the computations as being representative of actual compressor data.

The results of this Section and the previous one thus suggest that as rotor loading increases, the increase in tip region loss and endwall blockage influence the peak pressure rise capability of rotor examined.

3.3 Stator Flow Field

In addition to the rotor, it is also important to quantify the flow in the downstream stator since this determines the exit boundary condition for the upstream row. Since steady flow models are utilized in NiSTAR and NASTAR, the influence of the downstream stator is associated with the radial back pressure profile enforced at the rotor exit. NiSTAR utilizes non-reflective interface boundary conditions between the blade rows [64] where the circumferentially averaged radial profile upstream of the stator is specified. In contrast, NASTAR utilizes an axisymmetric bodyforce region downstream of the rotor to simulate the presence of the stator pressure field [49].

The stator pressure rise characteristics from the steady two-row simulations are shown in Figure (3.9). For all operating conditions, with either code and dense/coarse grids, the stator produces a strongly negatively sloped characteristic, indicating stalled operation. Again the NiSTAR coarse grid simulation yields the lowest pressure rise capability, and the NiSTAR dense rotor grid and NASTAR results are nearly identical. The
approximately constant difference (~5%) between the NiSTAR and dense grid NiSTAR stator pressure rise characteristics results from the differences in stator inlet conditions. The inlet profiles to the stator were obtained from the rotor exit flow, and as shown in Section (3.2.2), this is dependent upon the rotor computational grid. Figure (3.10) shows the pitch averaged stator inlet axial velocity profiles from the coarse and dense grid NiSTAR solutions at peak loading (points A and B). The stator inlet velocity defect in the tip region (outer ~10%) is larger with the coarse rotor grid, and the stator pressure rise difference is thus also related to the rotor tip flow resolution.

The stator loss buckets, given in Figure (3.11), illustrate that for bowed stators over the range of inlet flow angle considered, the loss remains nearly constant. The loss coefficient shown is defined as the absolute total pressure loss normalized by the difference between the mass averaged absolute total pressure and static pressure at inlet to the stator row (i.e. \( \Delta P_{t}/(P_{t,in} - P_{s,in}) \)). The flatness of the loss bucket is similar to the experimental data in Figure (2.5). In addition, the present stage operates at approximately 70% reaction, hence the rotor carries the majority of the pressure rise and loss. This along with the absence of a clearance flow allows the stator flow to maintain relatively low loss at all loading conditions.

The bowed stator design helps ensure that the stator endwall flow remains unseparated through development of radial pressure gradients in the blade passage which unload the endwall regions and increase stator midspan pressure rise. To establish that the stator is not limiting the pressure rise capability of the stage, the Diffusion Factor [54] computed at midspan, is given in Figure (3.12). The definition of Diffusion Factor is given by,

\[
DF = 1 - \frac{V_{exit}}{V_{in}} + \frac{\Delta V_{\theta}}{2 \sigma V_{in}}
\]

and is shown plotted against the stator loading for the NiSTAR and NASTAR simulations. Based on a considerable amount of experimental data, if the value of \(DF\) is below
approximately 0.6 the blades will remain unseparated [54, 71]. As shown in Figure (3.12) for all of the cases in this study this criteria is satisfied, and stator flow separation appears unlikely.

3.4 Overall Stage Performance

As a final evaluation of stage performance from the steady flow computations, the overall pressure rise characteristics are shown in Figure (3.13). Again for similar computational grids the NiSTAR and NASTAR simulations provide nearly identical pressure rise, while the coarse grid yields a slightly lower level.

The stage “stalling” or peak pressure rise coefficient\(^8\) predicted by NiSTAR and NASTAR is shown in Figure (3.14), along with experimental data from several low speed stages as reported by Koch [3]. Koch [3] utilized the blade diffusion length normalized by staggered spacing (analogous to length/width) as the correlating parameter, since for simple two-dimensional diffusers this parameter influences the pressure rise capability. As illustrated in Figure (3.14), the NiSTAR and NASTAR results are within the range of existing data, and thus the stage is representative of subsonic machines with high blade loading capability.

To illustrate the trend in overall stage performance, the normalized stage adiabatic efficiency from the computations is given in Figure (3.15). As compared to the NASTAR predictions, the NiSTAR simulations with a coarse grid have lower peak efficiency since the maximum rotor loss is approximately 15% above that given by NASTAR. For all cases the predicted efficiency trends are consistent with the findings presented for the rotor and stator and appear qualitatively similar to experimental data.

\(^8\) As defined by Koch [4], the effective pressure rise coefficient is an enthalpy-equivalent static pressure rise based on pitchline free-stream dynamic head.
3.5 Summary and Conclusion

The results of the steady flow simulations can be summarized as follows:

- Computed performance trends for the rotor and stator were realistic and consistent with existing experimental data.

- For similar computational grids, the rotor and stator flow fields were essentially independent of the code utilized.

- The averaging plane technique and bodyforce/deterministic stress models produced similar results for the steady flow environment examined here.

- Based on comparisons with NASTAR predictions and experimental data, the NiSTAR predictions with the baseline grid appear adequate for subsequent unsteady flow simulations to be performed.

- Stage pressure rise capability was limited by the rotor tip clearance flow, not by airfoil separation.

- Rotor endwall loss and blockage increased with blade loading, indicating rotor sensitivity to tip clearance.

- Flow structures in the rotor were similar despite differences in computational grid and CFD code.

- Flow within the stator remained attached over the range of stage operation examined.

- Differences in turbulence modeling did not significantly impact stage performance trends.
Figure (3.1): Rotor pressure rise characteristics

Figure (3.2): Rotor mass averaged relative total pressure loss buckets
Figure (3.3): Rotor inlet axial velocity profiles at operating points A, B, and C

Figure (3.4): Rotor pitch averaged spanwise loss profiles at operating points A, B, and C
(a) Rotor relative total pressure contours (contour increments = 10% of rotor inlet dynamic pressure)

(b) Static pressure contours (contour increments = 8% of rotor inlet dynamic pressure)

Figure (3.5): Axial planes at 2/3 rotor chord and tangential planes at the center of the rotor tip clearance gap. Operating points A, B, and C shown.
Figure (3.6): Rotor pitch averaged spanwise loss profiles at operating points A and D

Figure (3.7): Rotor entropy rise with increased blade loading
Figure (3.8): Rotor flow blockage variation (data from Khalsa [70])

Figure (3.9): Stator pressure rise characteristics
Figure (3.10): Stator inlet axial velocity profiles at peak pressure rise

Figure (3.11): Stator mass averaged absolute total pressure loss bucket
Figure (3.12): Stator midspan diffusion factor variation with increased blade loading

Figure (3.13): Stage pressure rise characteristics
Figure (3.14): Comparison of the peak effective static pressure rise coefficient with the stalling pressure rise data of Koch [3]

Figure (3.15): Stage mass averaged adiabatic efficiency variation
4.1 Introduction

In this chapter the impact of the downstream stator pressure field on the upstream rotor is examined in the unsteady flow environment. The primary objective is to identify and quantify the response of the rotor tip clearance flow. An additional goal is to determine how the time averaged rotor performance differs from that obtained using the steady flow approximations shown in Figure (2.2).

Results from two full interaction unsteady flow simulations of the baseline stage configuration are presented at conditions of peak pressure rise (high loading), and below design point pressure rise (low loading). These correspond to points A and D on Figure (3.13). The two cases are discussed in detail because they bracket the loading range of interest in modern HPC stage design. The flow fields encountered as well as the quantitative results obtained are thus representative of those obtained in subsequent unsteady flow computations given in the sensitivity study of Chapter 5.

Results from the two simulations are contrasted with the steady flow computations described in Chapter 3 and with steady flow simulations for a rotor subjected to an axisymmetric back pressure profile. Throughout the analysis emphasis is placed on quantifying the effects of unsteadiness on rotor endwall and tip clearance flow. Although the effect of unsteady rotor exit flow on the stator flow field and performance was not a focus of this study, some results of the computations are presented in Appendix B for completeness.
4.2 Rotor Flow Behavior

The unsteady flow computations were conducted with the NiSTAR code as described in Section (2.4.1). The full interaction flow fields were initialized with the converged steady flow solutions obtained using the averaging plane method. Thus, for a given operating condition the overall stage total-to-static pressure rise was kept the same in both the steady and unsteady flow simulations. The unsteady computations were run for several rotor-stator blade passing periods to establish periodicity in the primary flow variables. To attain convergence of the simulations, all of the cases in this investigation were run for a minimum of 10 cycles after clear periodic behavior was achieved. For the present study, the flow variables examined to establish and ensure periodicity (and numerical convergence) included rotor exit mass flow, rotor relative total pressure loss, rotor pressure and temperature ratios (efficiency), as well as static pressures at locations along the rotor trailing edge and downstream stator leading edge. These variables were chosen since the rotor flow field behavior is of primary interest in this investigation, and they convey much of the essential fluid dynamic information.

Rotor Exit Pressure Field

With regard to the rotor, what distinguishes the unsteady flow from the steady approximations is the presence of circumferential pressure non-uniformities imposed by the downstream stator. At a particular axial location behind the rotor, the amplitude of the circumferential variation depends upon the stator operating condition (Mach number and loading) and the geometric configuration (blade thickness and spacing). The frequency with which a rotor blade experiences the variations in back pressure is dependent upon the rotor wheel speed and the azimuthal spacing of the stator blades (i.e. blade passing
frequency). For the present cases a 1:1 rotor-stator blade count ratio was utilized so the
downstream stator row imposed forcing at the rotor passing frequency.\(^9\)

To indicate the level of pressure field non-uniformity seen at the rotor exit, the
circumferential pressure distribution at four spanwise locations is shown in Figure (4.1).
These results are for one representative instant of time, at the middle of the rotor-stator axial
gap, at the peak pressure rise operating point. The static pressure has been normalized by
the rotor outlet average dynamic pressure, and the peak-to-peak variation is approximately
20% of this quantity.

The radial profiles at five pitchwise stations at the same axial location and instant of
time are shown in Figure (4.2a), with the corresponding pitchwise average given in Figure
(4.2b). The individual radial profiles show a hub-to-tip pressure non-uniformity of
approximately 6 to 8% of the rotor exit dynamic pressure. Therefore, the circumferential
pressure non-uniformity is more than twice as large as the imposed radial pressure
variation. For the baseline stage examined this was true at both the high and low pressure
rise operating conditions.

The peak-to-peak amplitude of the circumferential pressure variation provides a
measure of the degree of rotor exit flow unsteadiness, and this will be utilized as a metric
for the overall level of unsteady forcing seen by the upstream rotor. Because the impact of
unsteady forcing on the rotor tip clearance vortex is also of interest, another metric is the
amplitude of circumferential pressure variation normalized by the dynamic pressure
averaged over the vortex region. (The latter will be larger than that based on the overall
rotor exit condition since the endwall region velocity defect yields a lower dynamic
pressure.) For the two operating conditions considered here, Table [4.1] provides these
parameters based on unsteady data obtained at the mid-axial gap location.

---

\(^9\) In Chapter 5 the effect of blade count ratio is interrogated and found to have a minor impact on results
presented here.
Table [4.1]: Rotor Exit Flow Field Parameters

<table>
<thead>
<tr>
<th></th>
<th>$P' / Q_{elu}$</th>
<th>$P' / Q_{vort}$</th>
<th>$M_x$</th>
<th>$M_y$</th>
</tr>
</thead>
<tbody>
<tr>
<td>High Loading (pt. A)</td>
<td>20%</td>
<td>38%</td>
<td>0.275</td>
<td>0.177</td>
</tr>
<tr>
<td>Low Loading (pt. D)</td>
<td>23%</td>
<td>41%</td>
<td>0.297</td>
<td>0.176</td>
</tr>
</tbody>
</table>

The increase in the unsteady forcing parameters with reduced stage loading was at first surprising, since one generally assumed that the amplitude of downstream unsteadiness is greater at peak blade loading. To understand why this need not be the case, consider steady, two-dimensional, isentropic compressible flow entering a cascade of stator blades. For such a flow, the linearized equation governing the behavior of pressure perturbations upstream of the blades is [72],

$$
\frac{1}{a^2} \left( \bar{V}_x \frac{\partial}{\partial x} + \bar{V}_y \frac{\partial}{\partial y} \right)^2 P' = \frac{\partial^2 P'}{\partial x^2} + \frac{\partial^2 P'}{\partial y^2}
$$

(4.1)

which has a general solution for evanescent pressure disturbances,

$$
P'_n(x, y) = A_n \exp \left\{ \frac{2\pi n}{s} \left( \left[ \frac{(1 - M_x^2 - M_y^2)^{1/2} + iM_x M_y}{1 - M_x^2} \right] x + i y \right) \right\}
$$

(4.2)

where

- $n =$ harmonic of disturbance
- $M_x = \bar{V}_x / \bar{a}$
- $M_y = \bar{V}_y / \bar{a}$
- $x < 0$ upstream of blade row

The unknown constant $A_n$ is set by the amplitude of the perturbation at $x = 0$, and $s$ is the stator blade tangential spacing. The pressure field decays upstream and is periodic in $y$ with multiples of the stator pitch. Since the first harmonic will have the greatest upstream influence, examine the magnitude ratio of $P'$ at $x$ and $x = 0$,  

75
\[ \frac{\|P'(x)\|}{\|P'(0)\|} = \exp \left\{ \frac{2\pi}{s} \frac{(1 - M^2 - M^2_x)^{\frac{1}{3}}}{(1 - M^2_x)} x \right\} \] (4.3)

Using equation (4.3) as a guide, consider the two cases listed in Table [4.1]. As the stage loading is increased, the absolute axial Mach number exiting the rotor (and entering the stator) decreases. In contrast, the absolute tangential Mach number remains nearly constant since there is little change in rotor flow deviation. Equation (4.3) indicates that the increase in axial Mach number at reduced loading will cause the stator pressure disturbances to decay more slowly which allows the pressure non-uniformity introduced at the stator row to be felt further upstream (an analogous result for the decay of disturbances upstream of a rotor is given in [73]). This does not, however, imply that the rotor response to exit unsteadiness will be attenuated at high loading, it only indicates that the level of downstream forcing was reduced as the through flow Mach number decreased.\(^{10}\)

### 4.2.1 Fluid Dynamic Features

The impact of unsteady downstream forcing on the rotor flow features will first be discussed qualitatively. Quantification of the unsteady flow effects on rotor performance is then provided in Section (4.3).

**Wake Behavior**

The interaction of rotor wakes with the downstream stator blades has been addressed by numerous other studies, and hence the influence of this interaction on the stator performance will not be detailed here.\(^{11}\)

Figure (4.3) shows entropy contours at midspan for four time instants during a single stator passing period. The snapshots are for the stage operating at the peak pressure

---

\(^{10}\) Additional discussion on the effects of back pressure amplitude variation is presented in Chapter 5 with regard to changes in rotor-stator axial spacing.

\(^{11}\) See Appendix B for results concerning the stator.
rise condition. The flow response is periodic at the blade passing frequency. As expected the rotor wakes are cut into distinct segments which then convect through the stator passages. The rotor midspan flow is well behaved with thin boundary layers and attached flow throughout the interaction period. The wake segments stretch, tilt, and diffuse as they pass through stator row similar to the behavior observed in the single row computations of Valkov [43] and the multi-row computations of Gundy-Burlet et al [44]. The wake sections appear to narrow along the stator suction surface, while spreading slightly on the pressure surface. This higher entropy fluid is retained near the pressure surface similar to the total temperature segregation process described by Kerrebrock and Mikolajczak [37]. In addition, throughout the interaction the stator midspan boundary layers remain attached and relatively thin. Shed vortices are also present in the downstream stator wakes. Therefore, the rotor wake behavior is at least qualitatively similar to that obtained by other investigations, and appears adequately captured in the present study. This observed flow response lends further confidence to the computations as providing a model which is representative of the unsteady flow environment.

**Tip Vortex and Endwall Flow Behavior**

The main objective of this study was to determine how downstream unsteadiness impacts the rotor tip vortex and endwall flow field. To address this, Figure (4.4) shows the rotor relative total pressure in an axial plane, at the mid axial gap location, at eight instants of time during two stator passing periods. Two adjacent rotor passages are illustrated.\(^\text{12}\)

In the first frame \((0.1\Delta t)\) the wake is well defined as is a small corner separation located near the suction surface-hub corner. The rotor-stator midspan and endwall axial alignment is as shown in Figures (4.3) and (B.1). The tip vortex is located at

\(^{12}\) Corresponding top views of the stage flow field showing interaction of the vortex with the stator are presented in Appendix B.
approximately mid-pitch and has a radial extent corresponding to roughly the outer 20% of span. In the second frame (0.4Δt) the vortex has moved radially downward and tangentially towards the pressure surface of the adjacent rotor blade, while the wake appears to thicken. The third frame (0.7Δt) shows that the vortex has intercepted the wake of the adjacent blade, with the core of the vortex located at approximately 80% span. This vortex-wake interaction indicates that the trajectory of the vortex has a tangential component such that at this downstream axial location, the wake and vortex meet. By the fourth snapshot (1.0Δt) the vortex is well within the wake and begins to diminish in size. The region of lower static pressure within the vortex core does persist since the section of wake just below the vortex appears to be drawn toward the core. Note that the remainder of the wake now resembles that at (0.1Δt). In the fifth frame (1.3Δt) the vortex appears somewhat combined with the wake and is beginning to reemerge from the suction side of the wake. In the sixth (1.6Δt) and seventh frames (1.8Δt), the vortex is no longer visible as a well defined structure but is rather a tangentially spread region of low relative total pressure at the upper end of the wake and along the casing. By the last frame (2.0Δt) the flow contains a well defined vortex which has grown in size from that at (1.8Δt).

The similarity between the (2.0Δt) frame and the (0.4Δt) image indicates that the fluid dynamic period required for the vortex to repeat its motion and interaction with the wake is approximately 1.6 to 1.8 stator passing periods. Although only two representative stator passing periods are shown in Figure (4.4), calculations over subsequent stator passings showed the vortex motion to repeat. Hence the behavior was periodic with a frequency different than the stator passing frequency. Additional video animations which were conducted of rotor exit flow field shown in Figure (4.4) utilized 20 snapshots and gave a vortex periodicity equal to 1.7 ± 0.1 stator passing periods.

As observed from Figure (4.4), there is both radial and tangential motion of the vortex from the rotor suction surface to the adjacent blade pressure surface. During this
process, the vortex collides with the wake and is difficult to identify. The vortex appears to pass through this plane as a region of continuously varying size as opposed to the situation observed in the steady flow simulations (Figure (3.5a)). The low swirl ratio associated with the vortex (swirl velocity of vortex core fluid/axial velocity < 0.6) also adds to the problem of visualizing it. At the rotor exit the vortex core has little static pressure defect (see Figure (3.5b)) and the endwall region resembles more of a velocity defect than a well defined, strongly swirling, vortical structure. As a result of the periodic motion, oscillations in endwall loss and blockage are to be expected.

Visualization of the unsteady rotor exit flow field for all of the cases considered in this investigation has shown that tip clearance vortex motion is always present and similar to that in Figure (4.4). The periodicity and motion of the tip vortex has not, to the authors knowledge, been previously reported upon and thus a complete explanation of this response has not yet been established. However, some observations regarding the vortex behavior and the resultant impact on rotor performance have been obtained, and these will be discussed in the next sections.

Tip Vortex Period

The periodicity of the rotor endwall flow at a frequency other than the stator forcing is characteristic of nonlinear systems where forcing at a given frequency may produce a response at another frequency [74, 75]. Nonlinear systems with frequency response characteristics which contain frequencies below the forcing frequency (as an example consider subharmonic response) are known to be those with nonlinear restoring forces. Hence it appears likely that a nonlinear fluid dynamic effect is responsible for the vortex behavior.

---

13 As an example, consider the well known mass-spring-damper system with sinusoidal forcing. For subharmonics to exist in this system, the spring need only have a nonlinear force-displacement characteristic of the form \( F = k (x + x') \). The spring force is the restoring force in this system. See [75] for details concerning this example.
As observed in Figures (4.3) and (4.4), the wake responds at the stator passing frequency while the tip vortex does not. The existence of various time scales in the rotor-stator flow field is not a new observation. Examples of flow phenomena which have time scales that do not depend on the stator passing period include rotor blade vortex shedding, the time required for secondary flow formation, and the time scale associated with boundary layer development. To determine what time scales were present in the stage flow field, Fast Fourier Transforms (FFT) of time traces from various locations in the computational domain were examined. As a check of solution integrity, the FFT's were utilized to show that standing waves, duct modes, and stationary acoustic disturbances were not present in the flow field. An additional indication of the vortex motion was also obtained by examining a time trace of static pressure taken on the downstream stator leading edge near the casing (97% span). This location provided a signal utilized to determine computational convergence, and as shown in Figure (B.1), it was also subject to the incoming rotor tip vortex. This time trace, which is shown in Figure (4.8a), indicates that the primary flow period is that associated with the rotor blade passing. However, the FFT of this signal given in Figure (4.8b) also indicates a disturbance at approximately one-half the rotor blade passing frequency. FFT's taken at locations downstream of the stator leading edge showed the decay of the amplitude of the 0.5 blade passing frequency disturbance. In addition, time traces at midspan did not indicate the presence of this frequency component. Thus the 0.5 blade passing frequency disturbance in stator leading edge static pressure seemed to correspond to the periodic movement of a flow structure over approximately two blade passing periods, which was relatively close to the observed vortex motion frequency.

The formation of the tip vortex is an inertial effect associated with the flow field, and the time scale associated with this is on the order of the rotor through flow (or convection) time. The correlation of the time associated with the tip vortex motion and the rotor flow through time is shown in Figure (4.9) for the two cases considered in this
Chapter. For the baseline stage configuration the vortex period was thus within approximately 10% of the rotor through flow time.

4.2.2 Time Averaging

Computational Approach

Before discussing the results of the present simulations, the issue of time averaging of flow variables in an unsteady flow environment must be addressed. Temporal averaging of primitive variables including $\rho, \rho u, \rho v, \rho w$, $T$ and $P$ can be accomplished using the standard definition,

$$\tilde{f}^t = \frac{1}{\Delta T} \int_T^{T+\Delta T} f(t) \, dt$$  \hspace{1cm} (4.4)

However because time averaging does not commute, many mathematical relations cannot be directly applied to the time averaged variables. For example,

$$\overline{(u^2)}^t \neq \overline{(\bar{u}')}^2$$  \hspace{1cm} (4.5)

$$\overline{p^t}^t \neq \overline{\bar{p}'^t}^t \left(1 + \frac{(\gamma - 1)}{2} \left(\overline{\bar{M}^t}^t\right)^2\right)^{\gamma/(\gamma - 1)}$$  \hspace{1cm} (4.6)

$$\overline{\bar{\omega}^t}^t \neq \frac{\overline{\bar{p}'^t_{r1}} - \overline{\bar{p}'^t_{r2}}}{\overline{\bar{p}'^t}}$$  \hspace{1cm} (4.7)

For unsteady flows relations such as (4.5) to (4.7) should be applied on an instantaneous basis. To utilize the time averaged variables appropriate higher order terms should be included in the equations (e.g. $\overline{(u^2)}^t = (\overline{\bar{u}'}^t)^2 + 2 \overline{\bar{u}'u'} + (\overline{u'}^t)^2$). The present investigation avoids this by utilizing time averages of the instantaneous parameters, hence,

$$\overline{\bar{p}'^t} = \frac{1}{\Delta T} \int_T^{T+\Delta T} P \left(1 + \frac{(\gamma - 1)}{2} \overline{\bar{M}^t}^t\right)^{\gamma/(\gamma - 1)} \, dt$$  \hspace{1cm} (4.8)
\[ \bar{\omega}' = \frac{1}{\Delta T} \int_{T}^{T+\Delta T} \left( \frac{P_{T_1} - P_{T_2}}{P_{T_1} - P_1} \right) dt \]  

(4.9)

where \( \Delta T \) is an appropriate fluid dynamic period based on the flow phenomena of interest. Therefore, all of the time averaged results were computed using equation (4.4).

**Results from the Present Investigation**

Initial thoughts regarding the unsteady flow within the stage were that the primary flow phenomena in the rotor passage would be locked to the stator passing period, and thus computation over a single stator passing cycle would be sufficient. From the images shown in Figure (4.4) this time scale is not appropriate for the rotor endwall flow field which has periodicity close to the blade through flow time. The longer period associated with the movement of tip vortex made it necessary to time average over a time scale appropriate for the endwall flow. In this case, the nearest integer time scale was two stator passing periods.\(^{14}\) Since it is frequently assumed that the flow in turbomachine rotors is phase locked to the blade passing period, it was of interest to determine how different the rotor exit flow would appear if it were time averaged over 1 and 2 stator passing periods respectively.

The time averaged rotor exit flow fields for 1 and 2 stator passing periods are shown in Figures (4.5) and (4.6). The 1-cycle average was done during the first stator passing period shown in Figure (4.4), when the vortex interacts with the wake of the adjacent blade. As a result, there is a larger region of low relative total pressure fluid in the time averaged endwall flow field shown in Figure (4.5). In contrast, the 2-cycle time average incorporates both the vortex-wake interaction and the period after this when the vortex structure is re-established. This average combines the first stator period in which there are higher endwall losses with the second stator passing period which has lower

\(^{14}\) Although the actual endwall flow periodicity is approximately 1.7 stator passings, time averaging over the next integer multiple of this (17), was not feasible given the computational resources.
endwall losses. Therefore, when compared with the 1-cycle case, the 2-cycle average has a somewhat smaller region of low total pressure fluid located near the casing. Outside the endwall region, the two averages appear nearly identical indicating that the wake/profile region of the flow field is periodic with the stator passing.

The difference in the endwall flow over the two periods implies that capturing an accurate time averaged picture of this portion of the flow requires knowledge of the fluid dynamic time scales occurring in the unsteady flow environment. In particular, blade passing phase locked averaging of the endwall flow appears inappropriate, and therefore in all subsequent analyses the 2-cycle average was utilized.

To show the difference between the time averaged and steady flow predictions, the result obtained from the steady averaging plane simulation of Chapter 3 is given in Figure (4.7). The steady flow has lower relative total pressure in the endwall region, and the vortex interacts only slightly with the upper end of the wake. The wake is thin and clearly defined over the remainder of the span. Contrasting this with the 2-cycle average of the unsteady flow simulation, the steady flow contains larger defects in relative total pressure in the outer 10-15% of span. The flow features in the 2-cycle average appear qualitatively more diffuse; the wake appears broader and the tip region lacks the organized structure found in the steady case. These differences are perhaps not surprising since the effects associated with tip vortex motion are not incorporated into any of the existing steady flow models. Further, due to the non-integer nature of the vortex passing period, the average passage method of Adamczyk [47] (on which NASTAR is based) will only capture unsteady phase locked flow phenomena and not the time averaged effects of the vortex motion. Similarly, the averaging plane method employed in NiSTAR was not formulated to capture this phenomena since circumferential averaging of the downstream stator pressure field eliminates the unsteady forcing on the rotor. In summary neither of the current steady flow models produced the qualitative tip vortex behavior observed in the time averaged unsteady flow simulations.

83
Summary of Observed Flow Behavior

Large scale tip vortex motion was observed in the rotor blade passage with frequency lower than the stator blade passing frequency. In contrast, the wake flow behavior was periodic at the stator passing frequency. The additional frequency associated with the endwall flow necessitates time averaging over a time scale different than stator passing, and this will influence the interpretation of quantitative results in the next section. Finally, existing steady models did not capture the time averaged flow field associated with the tip vortex motion in the unsteady flow computations.

4.3 Rotor Performance Characteristics

In this Section, quantitative comparisons will be made between the steady flow results of Chapter 3, and the two baseline unsteady simulations.

4.3.1 Pressure Rise Capability

The most important rotor performance quantity is the pressure rise-flow characteristic. Figure (4.10) compares the steady NiSTAR results described in Chapter 3, with the unsteady flow computations at high and low loading. The overall stage total-to-static pressure ratio was kept the same for both steady and unsteady flow simulations allowing a direct comparison. The instantaneous path taken during the two stator passing periods is shown as a dashed curve along with the time average. The pressure rise variation is approximately 3.5% of the time average at high loading, and 4.9% of the time average at low loading. This also illustrates that the rotor exit unsteady pressure fluctuation is slightly greater at the lower loading (higher through flow Mach number) operating condition. In contrast the mass flow variations are only 1% of the time average at both high and low loading points. The time averaged pressure rise and flow coefficients are within 1% of those obtained from the steady flow simulations.
Figure (4.11a) provides the steady and time averaged spanwise distributions of rotor loading at the high loading condition. The time averaged tip loading is slightly above that for the steady computation. The maximum difference between the two occurs in the outer ~10% of the span and is less than 5% of the steady value. The corresponding unsteady variation in spanwise loading is shown in Figure (4.11b) and the fluctuations are seen to be within approximately ±5% of the time mean. The loading oscillation appears smaller than that based on the exit dynamic pressure ($\Delta P'/Q_{exit}$) (given in Table 4.1) due to non-dimensionalization of the pressure rise by the inlet, as opposed to outlet, dynamic pressure.

In the present investigation the overall pressure rise and flow capacity predicted by the steady and time averaged unsteady simulations agreed. In general, however, the steady and time averaged results could differ, and therefore no universal conclusion should be drawn regarding the ability of current steady flow models to capture the pressure rise and flow capacity obtained in the unsteady flow environment. This can only be ascertained on a case-by-case basis.

4.3.2 Rotor with Axisymmetric Exit Condition

To determine the level of approximation needed to capture the overall results of the present study, a simple model based on representing the adjacent stator pressure field as an axisymmetric radial pressure profile applied downstream of the rotor, was examined. A schematic of this configuration is shown in Figure (2.2).

The axisymmetric profile used was obtained from the time averaged, pitch averaged, static pressure profile at the rotor-stator mid axial gap in the unsteady flow simulations. The back pressure profile was averaged over two stator passing periods. The radial pressure distribution obtained at high loading is shown in Figure (4.12). The spanwise variation is approximately 3% of the average rotor exit dynamic pressure (note
the similarity between this profile and the pitch averaged profile at one instant of time shown in Figure (4.2b)).

This pressure profile was applied to an isolated rotor with inlet and exit ducts extending one blade span. The original rotor computational grid (Section (2.4.1)) was modified to include an additional 45 axial grid points and planes downstream of the trailing edge with the exit static pressure profile imposed at the far downstream boundary. To account for the decay in the back pressure non-uniformity the variation in the imposed back pressure was slightly increased (while keeping the mean fixed) so that at the spatial location corresponding to the rotor-stator mid axial gap, the pressure distribution was approximately that given in Figure (4.12). Utilizing this approach the circumferential pressure non-uniformity at the rotor-stator mid axial gap location was less than 2% of the rotor outlet dynamic pressure. This was variation was found to be due to the presence of the rotor. In addition, the radial profile obtained at the rotor-stator mid axial gap location was within 1% of the targeted radial distribution. This method was adopted because initial computations with a more closely coupled reflective pressure boundary condition (see Saxer [64]) caused the rotor pressure rise to be over predicted as compared to the time averaged computations.\textsuperscript{15}

The above procedure was applied at both high and low loading conditions and the results were also shown in Figure (4.10). The pressure rise and flow coefficients were within approximately 1 to 2% of those predicted by the averaging plane and time averaged unsteady flow computations. In addition, the results from the axisymmetric boundary condition cases were also qualitatively similar to those obtained from the steady averaging plane computations. More generally, the circumferential averaging methods appeared successful at predicting the time averaged rotor back pressure profile, because the stator

\textsuperscript{15} Close coupling of the reflective pressure boundary condition produced an effective "impedance mismatch" between the rotor flow field and the prescribed exit condition.
remained free from large scale flow separations. Thus in the present case the stator pressure field could be adequately modeled using techniques based on pitchwise averaging.

To determine the effect of flow unsteadiness on other aerodynamic parameters, Tables [4.2] and [4.3] summarize some of the figures of merit. The averaging plane, time averaged, and unsteady variation values are expressed as a percentage of (not a percent difference from) those obtained in axisymmetric boundary condition cases.

**Table [4.2] Rotor Performance Parameters at High Loading**

<table>
<thead>
<tr>
<th>Axisymmetric End Condition</th>
<th>Averaging Plane (%)</th>
<th>Time Averaged (%)</th>
<th>Unsteady Variation (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure Rise Coefficient, $\psi$</td>
<td>0.376</td>
<td>100.2</td>
<td>100.1</td>
</tr>
<tr>
<td>Flow Coefficient, $\phi$</td>
<td>0.481</td>
<td>100.2</td>
<td>99.9</td>
</tr>
<tr>
<td>Loss Coefficient, $\bar{\omega}^m$</td>
<td>0.048</td>
<td>96.5</td>
<td>110.1</td>
</tr>
<tr>
<td>Endwall Blockage, $A_r/c$</td>
<td>1.348</td>
<td>94.1</td>
<td>70.1</td>
</tr>
</tbody>
</table>

**Table [4.3] Rotor Performance Parameters at Low Loading**

<table>
<thead>
<tr>
<th>Axisymmetric End Condition</th>
<th>Averaging Plane (%)</th>
<th>Time Averaged (%)</th>
<th>Unsteady Variation (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure Rise Coefficient, $\psi$</td>
<td>0.348</td>
<td>98.9</td>
<td>99.5</td>
</tr>
<tr>
<td>Flow Coefficient, $\phi$</td>
<td>0.511</td>
<td>99.8</td>
<td>99.8</td>
</tr>
<tr>
<td>Loss Coefficient, $\bar{\omega}^m$</td>
<td>0.043</td>
<td>102.0</td>
<td>103.9</td>
</tr>
<tr>
<td>Endwall Blockage, $A_r/c$</td>
<td>0.870</td>
<td>93.7</td>
<td>82.6</td>
</tr>
</tbody>
</table>

These tables illustrate that at both high and low loading the rotor loss and endwall blockage were primarily impacted by unsteady interaction with the downstream stator. Therefore the following two sections will examine the rotor loss and blockage behavior.

87
4.3.3 Aerodynamic Losses

As shown above the rotor aerodynamic loss is modulated by the rotor-stator interaction. The loss coefficients examined were based on changes in relative total pressure and entropy rise and were computed on an instantaneous basis using the methodology described in Section (3.2.3). The time averaged loss coefficients were then obtained by averaging the instantaneous values as discussed in Section (4.2.2). This approach was utilized because the inlet flow to the rotor was time invariant, and only the exit condition was changing in time.

The mass averaged rotor relative total pressure loss variation is shown in Figure (4.13) for the steady and unsteady flow simulations. At low loading, the time averaged loss is close to that obtained by the steady flow approximations. The amount of unsteady fluctuation is roughly ±20%, due to the interaction of the stator pressure field with the rotor at the elevated through flow Mach number. The unsteady fluctuation at high loading results in a time averaged loss 10% above that obtained using the averaging plane method. The difference between the time averaged and steady flow loss predictions decreases with reduced blade loading indicating dependence on both the fluctuation in back pressure, and the mean blade loading. Qualitatively, this may have been expected since the sensitivity of blade boundary layers and endwall flow regions to flow disturbances is amplified as the peak of the pressure rise characteristic is approached. It should be emphasized that the instantaneous loss variations are not caused by changes in rotor inflow angle since these are small (< 1% of rotor turning), and are not quasi-steady since the locus varies off the steady flow loss curve. The axisymmetric boundary condition model was also unable to yield the computed time averaged loss at high loading. Thus the difference in overall rotor loss predicted at near peak loading was not obtained by either of the steady flow models.

To determine which portions of the rotor flow field produce the additional time averaged loss at high loading, spanwise distributions of pitch averaged loss coefficient are given in Figure (4.14). Figures (4.14a) and (4.14b) provide comparisons of the loss
profiles obtained from the time averaged unsteady flow simulation, the axisymmetric exit condition simulation, and averaging plane computation described in Chapter 3. The axisymmetric and averaging plane cases have similar distributions of loss, with the axisymmetric prediction having slightly higher loss throughout. For both of the steady flow approximations the loss associated with the tip vortex region is confined to the outer 15% of span. Compared with the time averaged flow, the steady cases show lower loss along the blade ("profile" region) and higher loss in the tip region. The time averaged results do not contain as distinct a region of high loss fluid corresponding to the tip vortex. Instead, the loss associated with the tip region extends over the outer ~20% of span and is slightly below that obtained in the steady flow cases.

To determine why the time averaged loss profile appears as it does, spanwise loss distributions for 20 instants during two stator passing periods are shown in Figure (4.14c). The inner 75% of the span has loss fluctuations on the order of 50% of the mean. The outer 25% of span shows not only fluctuations but radial variations indicative of the spanwise movement of a region of high entropy fluid corresponding to the motion of the tip vortex observed in Figure (4.4). This reduces the time averaged depth of the relative total pressure defect in the endwall region. The unsteady response of the blade boundary layers, however, results in a higher level of time averaged loss, qualitatively similar to the behavior observed by Fritsch [76] for turbine rotor boundary layers subjected to the pressure field of an upstream nozzle guide vane. Compared to steady predictions, the time averaged profile loss is higher than that found in [76], and this may be due to the adverse pressure gradient in the compressor. There is little published data on loss generation in turbulent, diffusing, boundary layers with unsteady forcing at frequencies characteristic of blade row interaction, and it is difficult to ascertain if the modulation in profile loss is representative of that observed in the actual flow environment. What can be concluded is that the unsteady interaction has altered the overall distribution of loss such that, relative to the steady flow calculations, there are higher losses in the tip region and lower losses over the remainder of
the blade. For the present investigation, this was found to be true in all unsteady flow computations.

Because the motion of the rotor tip vortex appears linked to changes in rotor endwall loss, it is of interest to determine how the rotor tip region entropy rise varies. Figure (4.15) shows the overall, tip region, and midspan entropy rise fluctuations over two stator passing periods computed using the approach outlined in Section (3.2.3). The variations shown were obtained after approximately 18 stator passing periods when the unsteady solution was well converged. The rotor midspan time trace contains two peaks which occur at 20 - 40% and 120 - 140% of blade passing and reflect the instants when the rotor and stator were circumferentially aligned. This stator passing frequency response agrees with the wake behavior shown in Figure (4.3). The entropy rise in the tip region (outer ~20% of span) has a periodicity of approximately 1.7 ± 0.1 stator passing periods which corresponds to the observed flow behavior.\textsuperscript{16} Therefore, the overall rotor entropy rise also varies at a frequency different than blade passing.

In terms of amplitude of the entropy rise variations, the tip region oscillation is nearly twice as large as that observed at midspan. However, because approximately 70% of the overall entropy rise is due to blade boundary layer flow, the peak-to-peak amplitude of the overall loss variation is more strongly influenced by the midspan than the tip region.

To illustrate the difference between the steady and unsteady flow loss predictions, the minimum, maximum, and time averaged rotor entropy rise are given in Figure (4.16), plotted against the circumferential exit pressure variation normalized by the rotor exit average dynamic pressure (see Table [5.1]). The entropy rise is normalized by that obtained from the axisymmetric exit condition simulation at the same average loading. Figure (4.16) indicates that the predicted increase in overall rotor loss depends upon operating point as well as level of back pressure fluctuation. The time average does not

\textsuperscript{16} The computational resources available allowed the stage flow field to be saved at 20 instants during 2 stator passing periods. Thus the temporal resolution of the unsteady flow results is at best ±0.1 of a blade passing period.
occur mid-way between the minimum and maximum entropy rise indicating that the unsteady loss variation is not symmetric in time, as was also seen in the time traces of Figure (4.15).

Both steady and time averaged unsteady flow predictions show an increase in rotor loss with increased loading. For the averaging plane computations, Figure (3.7) showed a difference in overall entropy rise from low to high loading of approximately 10%. The difference in time averaged entropy rise between these two conditions is nearly 35%. The impact of rotor loading on loss is thus more pronounced in the unsteady flow predictions.

*Tip Region Loss*

If the rotor loss is separated to examine only that associated with the tip region (outer ~20% span), the results are quite different than for the overall rotor. Figure (4.17) shows rotor tip region entropy rise normalized by that obtained in the axisymmetric exit condition case (see Section (3.2.3) for methodology). The time averaged endwall loss is 10 to 20% below that in the steady flow computations, and again appears dependent on mean loading as well as the level of unsteady forcing. Examining the endwall flow field and the entropy rise variations, it was found that the maximum tip region entropy rise occurred when the tip vortex merged with the wake of the adjacent rotor blade.

Figure (4.17) also indicates that a difference exists between the rotor tip region loss predicted by the steady and unsteady flow simulations. To determine if the difference could be explained on a quasi-steady basis, a control volume analysis was utilized.

An incompressible, two-dimensional, control volume analysis similar to that of Storer and Cumpsty [18] was applied to the unsteady and axisymmetric exit condition cases at high loading.\(^\text{17}\) Such a description can be utilized to obtain the production of entropy, or

\(^\text{17}\) The effects of compressibility were ignored in this discussion to maintain clarity. The extension of the control volume model to compressible flow can be found in Denton [73]. Application of the compressible flow model to the steady and unsteady flow simulations yielded loss predictions which were within 3% of the incompressible results.
equivalently the loss in stagnation pressure associated with mixing of the tip leakage flow, since this is determined by overall conservation constraints. The model utilized is for a jet entering a uniform flow at an angle and mixing at constant area to a uniform state. To apply the model to the flow within the rotor, the leakage flow over the blade tip corresponds to the jet, while the remaining passage flow represents the uniform main stream. It can be shown that if skin friction effects are neglected the average total pressure loss coefficient is given by [18],

$$\bar{\omega} = \frac{\Delta P_{vel}}{\frac{1}{2} \rho V_1^2} = \chi \sin \alpha_j \left[ \frac{2 + \chi \sin \alpha_j - 2 \cos \alpha_j}{(1 + \chi \sin \alpha_j)^2} \right] \frac{\cos^2 \beta_1}{\cos^2 \beta_2}$$  \hspace{1cm} (4.10)

with

$$\chi = \frac{m_j}{m_1}$$  \hspace{1cm} (4.11)

where (1) and (2) denote the inlet to the main flow and the exit state respectively, and (j) denotes the clearance jet. The jet angle ($\alpha_j$) is based on a mass average of the leakage flow angle along the rotor chord. In deriving equation (4.10) the inlet total pressure and velocity of the two streams has been assumed to be equal.

Since equation (4.10) is primarily dependent on rotor tip leakage mass flux, the “over-tip” mass flow is shown in Figure (4.18) for the unsteady and axisymmetric exit condition cases. The main stream mass flux is approximately the same in both cases (as evidenced by the flow coefficient shown in Figure (4.10)), and the mass averaged leakage flow angles were within $\approx 3$ degrees of each other. As with the rotor tip region entropy rise variation shown in Figure (4.15), the clearance mass flux modulates through one cycle during approximately two stator passing periods. The magnitude of this oscillation is $\pm 11\%$ of the time average, and the time average being 6% below the axisymmetric exit condition value.

To determine how the leakage mass flux difference affects the tip region loss, Figure (4.19) shows the loss coefficient computed by applying the control volume model to
the axisymmetric exit case and to the unsteady flow simulation. For the unsteady flow, the
time averaged loss was calculated in two ways: (1) using the time averaged flow variables,
and (2) by time averaging the instantaneous loss values. This was done to see if the time
average loss coefficient would be greatly affected by the method of application of the
control volume analysis. As shown in Figure (4.19), using either method, the estimated
time averaged tip region loss was approximately 30 to 40% below that for the steady case.

Since the tip region loss obtained using the control volume analysis has a trend
similar to that observed in the computations, it was of interest to quantitatively compare the
loss computed using the control volume model with that calculated by mass averaging the
rotor tip region relative total pressure loss. Table [4.4] provides the values calculated from
the analysis and from the mid axial gap location in the computations.

Table [4.4]: Predicted Rotor Tip Loss at High Loading

<table>
<thead>
<tr>
<th></th>
<th>( \bar{\omega}_{tip} )</th>
<th>( \bar{\omega}_{CV} )</th>
<th>( \frac{\bar{\omega}<em>{CV}}{\bar{\omega}</em>{tip}} - 1 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axisymmetric Case</td>
<td>0.0144</td>
<td>0.0181</td>
<td>25.7</td>
</tr>
<tr>
<td>Time Average of</td>
<td>0.0115</td>
<td>0.0135</td>
<td>17.4</td>
</tr>
<tr>
<td>Unsteady Case</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

In both cases the control volume model predicts higher tip region losses, since it gives the
fully mixed out loss and the mid axial gap flow is not entirely mixed. The time averaged
case does however, show a difference of only 17.4% indicating that the flow is closer to
being fully mixed. This seems plausible since the motion of the tip vortex in the unsteady
flow environment decreases the depth of the time averaged relative total pressure defect in
the endwall region (as shown in Figure (4.14)). Therefore, reducing the tip mass flux
decreases loss as indicated by the control volume analysis and the computations, and the
additional "mixing" in the unsteady flow causes the time averaged tip region loss to be
closer to the fully mixed out prediction.
Connection to Rotor Efficiency

The entropy rise within the rotor can be related to the rotor efficiency as follows. For a compression process from an initial state (1) to a final state (2), the adiabatic efficiency is,

\[
\eta_c = \frac{\text{ideal work}}{\text{actual work}} = \frac{h_{2,i} - h_{1,i}}{h_2 - h_1}
\]  
(4.12)

or,

\[
\eta_c = 1 - \frac{T_{1,i} \Delta \xi}{h_2 - h_1}
\]  
(4.13)

where the subscript \(s\) denotes compression to an isentropic state having the same pressure as the actual (entropic) compression.

Equation (4.13) connects the difference in the predicted overall and rotor tip region entropy rise from the steady and unsteady flow simulations to the machine performance. Since the axisymmetric exit cases were formulated to provide the rotor with the same average pressure rise as the corresponding time averaged flow, the overall enthalpy rise and exit total temperature were also approximately the same as in the time averaged case. Therefore, the computed entropy rise will primarily determine the difference in efficiency between steady and unsteady flow predictions. Table [4.5] gives the ratios of efficiency calculated from the axisymmetric exit cases and unsteady flow simulations at high and low loading. The efficiency ratios shown are based on the overall rotor entropy rise and that occurring only in the tip region.

Table [4.5]: Rotor Efficiency Ratios

|                | Overall \(|\eta_{\text{uni}}/\eta_{\text{axi}}-1|\) (%) | Tip Region \(|\eta_{\text{uni}}/\eta_{\text{axi}}-1|\) (%) |
|----------------|-----------------------------------------------|-----------------------------------------------|
| High Loading   | -2.0                                          | 5.5                                           |
| Low Loading    | -1.5                                          | 1.9                                           |

Compared to the steady flow model, the unsteady flow computations show a reduction in overall rotor efficiency and an increase in efficiency in the tip region. This trend suggests
that the time averaged distribution of loss is not adequately captured by the steady flow simulations.

Summary

The time averaged unsteady flow computations showed an increase in overall rotor loss and a decrease in the tip region loss as compared to the steady flow simulations at the high and low rotor loading. The difference in tip region loss was found to be attributable to a difference in rotor tip leakage mass flux, and the tip vortex motion observed in the unsteady flow simulations.

4.3.4 Aerodynamic Endwall Blockage

The impact of the stator pressure field on the rotor endwall blockage is also of interest since for steady flows the blockage has been associated with the rotor pressure rise capability [19, 70]. As discussed in Section (3.2.4), the rate of growth of blockage in the tip region can be regarded as a metric to evaluate sensitivity to tip clearance. In Chapter 3, the blockage was quantified for the steady averaging plane computations and the results were found to be in the range of the experimental data reported by Khalsa [70]. The quantification of the endwall blockage in an unsteady flow is now addressed using the approach outlined in Section (3.2.4). For the unsteady cases the blocked area and the tip region loading parameter were computed in a quasi-steady manner and the time average was calculated by averaging the instantaneous values.

The results from the unsteady flow simulations at high and low loading are shown in Figure (4.20) using blockage and tip loading parameters computed at the rotor-stator mid axial gap. The unsteady flow results are plotted with those from the steady flow computations using the averaging plane and the axisymmetric exit condition, as well as the data of Khalsa [70]. At high loading, the instantaneous values vary by +90% to -40% of
the time average. The oscillation is due to the motion and interaction of the tip vortex with
the wake of the adjacent rotor blade.

As shown in Section (4.2.2), the time averaged flow exhibits lower endwall losses
than the steady flow approximations because of mixing due to vortex motion and a
reduction in the time averaged tip leakage mass flux. Both these effects tend to decrease the
magnitude of the tip region relative total pressure defect at the rotor exit and the associated
velocity non-uniformity. Thus, it does not seem surprising that the time averaged endwall
blockage is also below that obtained from the steady flow calculations.

The instantaneous blockage, however, can be greater than that found in the steady
flow as seen by examining the temporal variation in blockage at high loading shown in
Figure (4.21). The period of the blockage oscillation is approximately 1.7 ± 0.1 stator
blade passings. Similar to the rotor tip region entropy rise shown in Figure (4.15), the
endwall blockage attains a maximum when the tip vortex merged with the adjacent rotor
blade wake. This also corresponds to the qualitative flow behavior observed in rotor
relative total pressure contours shown in Figure (4.4). The variation in the tip loading
parameter shown in Figure (4.20) is primarily due to fluctuations in rotor tip region relative
total pressure loss (Figure (4.14c)), and not to oscillations in static pressure rise which are
only ±5% of the mean (Figure (4.11b)).

The oscillations in blockage and tip loading parameter indicate that for the unsteady
flow environment the instantaneous values are difficult to interpret. Therefore, with regard
computations, it is recommended that time averages over a sufficient number of blade
passing periods be utilized when calculating the endwall blockage, and that the data not be
phase locked. This will ensure that the time scales associated with endwall flow are
captured and that the blockage values are representative of the actual flow behavior.
4.4 Observations from Unsteady Flow Simulations

Controlled numerical experiments on unsteady rotor-stator interaction have shown that downstream unsteadiness introduced by the adjacent stator pressure field affects rotor flow phenomena as well as the predicted time averaged performance. The following list summarizes the observations.

**Observed Flow Features**

- Tip vortex motion on the scale of the rotor pitch was observed at the rotor exit.
- The period of the vortex motion scaled with rotor flow through time, not stator passing period.
- Tip vortex merged with the wake of the adjacent rotor blade at the rotor exit, during a portion of the vortex motion cycle.
- Due to tip vortex motion, the time averaged endwall flow field was not captured by the steady flow approximations examined.
- The regions of the flow field which are approximately two-dimensional (e.g. midspan) showed rotor-stator wake interaction similar to that found in other investigations.

**Performance Implications**

- Time averaging of performance parameters must be done over the tip vortex flow time scale.
- Compared to the steady flow models, the unsteady flow simulations showed:
  - little effect on overall rotor pressure rise and flow capacity
  - 10 to 20% decrease in time averaged rotor tip loss
  - 20 to 35% increase in time averaged rotor loss away from tip region
  - a radial distribution of rotor loss qualitatively different than that given by the steady flow computations (the time average had lower tip loss and higher profile loss, while steady cases had higher tip loss and lower profile loss)
  - large oscillations in instantaneous tip loss and endwall blockage due to periodic merging of tip vortex with the adjacent rotor blade wake
  - 10 to 40% decrease in time averaged rotor endwall blockage compared to the steady flow computations
  - effects of unsteadiness were more pronounced at high loading
4.5 Questions to be Addressed by Sensitivity Study

Based on the observations, several questions emerged regarding the behavior of the rotor flow field in the unsteady flow environment. It was determined that a sensitivity study would be appropriate since it allowed the effects of downstream unsteadiness to be interrogated over a range of parameters of interest for modern HPC stages. The list of questions to be addressed is as follows:

- How do:
  - stator radial design (i.e. radial distribution of back pressure)
  - rotor-stator axial gap (i.e. amplitude of back pressure variation)
  - rotor-stator blade count ratio (i.e. frequency of back pressure variation)

  influence the stage performance results?

- Are the unsteady flow phenomena qualitatively similar in all these cases?

- Is the unsteady interaction of the rotor with the stator pressure field truly bilateral, or can the stator response be regarded as an essentially constant amplitude sinusoidal variation in back pressure?
  (i.e. How much of rotor-stator interaction is one-way?)

- Is the observed behavior of the rotor tip region flow field caused by forcing from the downstream stator row, or is it self induced/self excited?
  (i.e. Would an isolated rotor have such a response?)

A controlled set of unsteady flow computations was conducted to answer these questions, and the results are presented in the following Chapter.
Figure (4.1): Circumferential pressure variation imposed on the rotor by the downstream stator at one instant of time (at peak pressure rise)

Figure (4.2a) Radial pressure variation imposed on the rotor by the downstream stator at one instant of time (at peak pressure rise)
Figure (4.2b): Pitch averaged radial pressure profile imposed on the rotor by the downstream stator at one instant of time (at peak pressure rise)

Figure (4.3): Entropy contours at midspan at 4 instants of time showing periodic interaction of the rotor wake with the downstream stator (contour increments = 20% of stage time averaged entropy rise)
Figure (4.3): - Continued. (contour increments = 20% of stage time averaged entropy rise)
Figure (4.4): Rotor relative total pressure contours showing motion of the tip vortex in an axial plane mid-way between the rotor and stator (contour increments = 5% of rotor inlet dynamic pressure)
Figure (4.4): - Continued. Tip vortex flow is approximately periodic over 2 blade passing cycles (contour increments = 20% of stage time averaged entropy rise)
Figure (4.5): Rotor relative total pressure contours time averaged over 1 blade passing period (in an axial plane mid-way between the rotor and stator; contour increments = 5% of rotor inlet dynamic pressure)

Figure (4.6): Rotor relative total pressure contours time averaged over 2 blade passing periods (in an axial plane mid-way between the rotor and stator; contour increments = 5% of rotor inlet dynamic pressure)

Figure (4.7): Rotor relative total pressure contours obtained from steady flow averaging plane simulation (in an axial plane mid-way between the rotor and stator; contour increments = 5% of rotor inlet dynamic pressure)
Figure (4.8): Stator leading edge static pressure time trace and corresponding FFT

Figure (4.9): Correlation of rotor tip vortex periodicity with rotor flow through time
Figure (4.10): Rotor pressure rise characteristic

(a) Averaging plane and time averaged results

Figure (4.11): Rotor spanwise loading distribution at peak pressure rise
(b) Instantaneous unsteady variation

Figure (4.11): Continued. Rotor spanwise loading distribution at peak pressure rise

Figure (4.12): Rotor back pressure profile obtained from the time averaged results and specified for the axisymmetric B.C. simulation at high loading
Figure (4.13): Rotor mass averaged relative total pressure loss bucket

(a) Averaging plane and time averaged results

Figure (4.14): Rotor pitch averaged spanwise loss profiles at high loading
(b) Axisymmetric B.C. and time averaged results

(c) Instantaneous unsteady variation

Figure (4.14): Continued. Rotor pitch averaged spanwise loss profiles at high loading
Figure (4.15): Rotor mass averaged entropy rise variation

Figure (4.16): Maximum, time averaged, and minimum mass averaged entropy rise for the rotor at high and low loading
Figure (4.17): Maximum, time averaged, and minimum mass averaged entropy rise for the rotor tip region at high and low loading

Figure (4.18): Rotor tip clearance mass flow variation
Figure 4.19: Rotor tip clearance loss computed using the control volume model

Figure 4.20: Rotor tip region flow blockage correlation (data from Khalsa [70])
Figure (4.21): Rotor tip region flow blockage variation
Chapter 5

Sensitivity Study on Unsteady Flow Effects

5.1 Introduction

In this chapter the results of a sensitivity study for the rotor and stage in the unsteady flow environment are presented. The objective was to evaluate the generality of the baseline results obtained in Chapter 4 and address the questions posed in Section (4.5). The following comparisons of time accurate flow simulations were conducted:

(1) Effect of stator radial geometry distribution

- Comparison of upstream rotor flow field with a conventional radial (straight) stator and with the non-radial bowed stator.

(2) Effect of rotor-stator axial spacing

- Comparison of upstream rotor flow field with a conventional straight stator located at half the nominal axial spacing, to that obtained with nominal axial spacing.

(3) Effect of rotor-stator blade count ratio

- Comparison of upstream rotor flow field with the bowed stator having a 2 rotor: 3 stator count ratio, to that obtained with a 1 rotor: 1 stator count ratio.

(4) Representation of the stator pressure field by a circumferentially varying, time invariant, exit boundary condition.

- Comparison of upstream rotor flow field with a circumferentially varying, time invariant exit pressure profile, to that obtained in the full rotor-stator calculation.

(5) Self-induced effects leading to fluid dynamic phenomena having time scale not equal to stator passing period.

- Examination of the time accurate simulation of an isolated rotor.

114
Because the results of Chapter 4 indicated that the unsteady effects were most pronounced at peak stage loading, all simulations were conducted at this operating condition, with the same time averaged stage total-to-static pressure rise. For the isolated rotor case, the time averaged blade row total-to-static pressure rise was held at a value corresponding to the full stage simulation.

5.2 Rotor Flow Behavior

As discussed in Section (4.2), one metric which can be utilized to characterize the unsteady flow simulations is the level of rotor exit circumferential pressure non-uniformity ($P'$). The azimuthal back pressure variations imposed by the stator row on the rotor were typically greater than the imposed radial variations, (as shown in Section (4.2)) so the quantities $P'/Q_{exit}$ and $P'/Q_{vort}$ provide measures of the unsteady forcing on the overall rotor flow field and the tip vortex. (These parameters do not, however, determine the sensitivity of the rotor to the forcing, i.e. the “output”.) Table [5.1] summarizes the forcing present at the rotor-stator mid-axial gap location in each cases examined.

Table [5.1]: Rotor Exit Flow Field Parameters

<table>
<thead>
<tr>
<th></th>
<th>$P'/Q_{exit}$ (%)</th>
<th>$P'/Q_{vort}$ (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotor-bowed stator, nominal case</td>
<td>20</td>
<td>38</td>
</tr>
<tr>
<td>Rotor-bowed stator, 2:3 count</td>
<td>15</td>
<td>28</td>
</tr>
<tr>
<td>Rotor-straight stator</td>
<td>18</td>
<td>37</td>
</tr>
<tr>
<td>Rotor-straight stator, 50% axial gap</td>
<td>42</td>
<td>81</td>
</tr>
<tr>
<td>Rotor alone with circumferential B.C.</td>
<td>21</td>
<td>28</td>
</tr>
<tr>
<td>Rotor alone with axisymmetric B.C.</td>
<td>“0”</td>
<td>“0”</td>
</tr>
</tbody>
</table>

The rotor-straight stator configuration has approximately the same degree of exit flow unsteadiness as the original rotor-bowed stator case on which it was based. The case with
a 50% reduction in axial spacing has a forcing almost a factor of two larger than the nominal configuration.\textsuperscript{18} (The dependence of rotor exit forcing on the blade row axial spacing (upstream distance, $x$) can also be seen in equation (4.2).) For the 2 rotor-3 stator case the amplitude of the forcing was reduced relative to the 1:1 blade count cases because the individual stator blades were scaled to maintain the same solidity as the original row (see Section (5.3.4)). The rotor with the circumferentially specified exit pressure profile also had approximately the same level of forcing as the baseline rotor-bowed stator case upon which it was based. Finally, the time accurate simulation of the rotor alone did not have continuous forcing applied to it; instead, an exit pressure pulse was utilized to determine the flow field response to non-periodic excitation (see Section (5.5)).

In each of the cases the unsteady and time averaged rotor performance parameters were computed and analyzed. Video animations of the unsteady flow field were also performed to identify the response of the key flow features within the rotor. Although the quantitative details were different, the flow behavior was in all cases qualitatively similar to that observed in the baseline simulations of Chapter 4.

\textit{Wake Behavior}

For all of the rotor-stator cases considered, the interaction of the rotor midspan wakes with the downstream blade row was like that shown in Figure (4.3). The only notable difference was in the 50% axial gap case where there was little mixing of the rotor wakes prior to entry into the stator row. As a result, the maximum depth of the wake velocity defect at the stator inlet plane was roughly 30% greater than that in the nominal spacing case.\textsuperscript{19} This increased rotor wake velocity defect produced a larger slip velocity in the absolute frame and more rapid migration of the wake segments toward the stator

\textsuperscript{18} Reducing the axial spacing by half moved the location of the rotor-stator mid-axial gap to 8% of rotor chord.

\textsuperscript{19} The rate of rotor wake decay with axial spacing was similar to the experimental data given in Gostelow [77].
pressure surface. The variation in stator inlet flow angle away from the endwalls (e.g. midspan) was also greater than in other cases. This increased the stator loading fluctuations to approximately 18% of the time averaged value.\textsuperscript{20}

Finally, for all of the unsteady flows simulations the interaction of the rotor wake with the stator blade row was at the stator passing frequency. This behavior is a kinematic consequence of the relative motion of the airfoil rows which is not qualitatively different for the aerodynamic and geometric parameters considered.

\textit{Tip Vortex Behavior}

As shown in Section (4.2.1), the rotor tip vortex exhibited radial and tangential motion in the unsteady flow environment with a period close to the rotor flow through time. A goal of the sensitivity study was to establish whether this response was specific to the geometric design and aerodynamic conditions considered in Chapter 4.

The results of the study were that tip vortex motion with a period near the rotor flow through time existed in all cases examined. (As will be discussed in Section (5.5), an attempt to self-induce or self-excite this behavior was unsuccessful.) Qualitatively, the vortex behavior was similar to that shown in Figure (4.4), where the vortex moved from near the rotor suction surface towards the adjacent rotor blade pressure surface. The periodic merging and re-emerging of the vortex with the adjacent rotor blade wake was also observed in all cases.

The correlation of the tip vortex period with the rotor flow through time is shown in Figure (5.1) for all of the unsteady flow simulations. The unit of time is the stator passing period since this is the characteristic time associated with the downstream forcing. As in Chapter 4, the instantaneous and time averaged performance parameters were computed using the nearest integer multiple of rotor passing periods corresponding with to the

\textsuperscript{20} See Appendix B for results concerning the change in stator flow field response.
endwall flow time scale (e.g. if the vortex period was 1.7 stator passing periods, and the simulation and flow field interrogation were conducted over 2 stator passing periods). The times associated with the 2:3 blade count case are therefore approximately 3/2 times as long as those obtained in the 1:1 cases. (If the normalization were performed using the rotor passing period, the 2:3 case frequencies would be 1.6, which is similar to that obtained in the 1:1 cases.) For each of the cases, the vortex period was determined using video animations of the rotor exit flow field and by spectral analysis (i.e. FFT's) of static pressure traces obtained from the downstream stator leading edge. The unsteady flow computations demonstrated that the behavior of the rotor flow field was neither steady in the relative frame nor phase locked to the passing of the adjacent blade row.

5.3 Rotor Performance Characteristics

In this Section, quantitative comparisons will be made between the steady flow results of Chapter 3, the axisymmetric exit condition cases, the baseline unsteady flow simulations of Chapter 4, and the unsteady flow results. As a summary, Tables [5.2] and [5.3] list the time averaged performance parameters and the level of fluctuation in each. The values have been expressed as a percentage of the axisymmetric boundary condition results at peak loading. In all cases the stage total-to-static pressure ratio is the same.

Table [5.2]: Time Averaged Rotor Performance Metrics

<table>
<thead>
<tr>
<th>Axial Coefficient, ψ</th>
<th>Flow Coefficient, φ</th>
<th>Loss Coefficient, ( \bar{\omega}^{in} )</th>
<th>Endwall Blockage, ( A_e/w_e )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axisymmetric Exit Condition</td>
<td>0.376</td>
<td>0.481</td>
<td>0.048</td>
</tr>
<tr>
<td>R-Bowed Stator</td>
<td>100.1%</td>
<td>99.9%</td>
<td>110.1%</td>
</tr>
<tr>
<td>R-Straight Stator</td>
<td>99.1</td>
<td>100.0</td>
<td>104.1</td>
</tr>
<tr>
<td>R-Straight Stator, 50% gap</td>
<td>101.4</td>
<td>100.2</td>
<td>114.9</td>
</tr>
<tr>
<td>2R-3 Bowed Stator</td>
<td>100.1</td>
<td>99.9</td>
<td>101.0</td>
</tr>
<tr>
<td>R-with circ. specified B.C.</td>
<td>100.6</td>
<td>99.8</td>
<td>106.0</td>
</tr>
</tbody>
</table>

118
Table [5.3]: Fluctuation in Rotor Performance Metrics

<table>
<thead>
<tr>
<th>Axisymmetric Exit Condition</th>
<th>Pressure Rise Coefficient, $\psi$</th>
<th>Flow Coefficient, $\phi$</th>
<th>Loss Coefficient, $\bar{m}$</th>
<th>Endwall Blockage, $A_e/\tau$</th>
</tr>
</thead>
<tbody>
<tr>
<td>R-Bowed Stator</td>
<td>98 to 102%</td>
<td>99 to 100%</td>
<td>87 to 128%</td>
<td>26 to 135%</td>
</tr>
<tr>
<td>R-Straight Stator</td>
<td>99 to 102</td>
<td>100 to 101</td>
<td>91 to 116</td>
<td>37 to 87</td>
</tr>
<tr>
<td>R-Straight Stator, 50% gap</td>
<td>97 to 103</td>
<td>99 to 101</td>
<td>67 to 144</td>
<td>27 to 103</td>
</tr>
<tr>
<td>2R-3 Bowed Stator</td>
<td>99 to 100</td>
<td>99 to 100</td>
<td>95 to 106</td>
<td>70 to 98</td>
</tr>
<tr>
<td>R-with circ. specified B.C.</td>
<td>100 to 101</td>
<td>99 to 100</td>
<td>84 to 122</td>
<td>33 to 133</td>
</tr>
</tbody>
</table>

Compared to the axisymmetric exit case, the differences in time averaged rotor loss and endwall blockage are much larger than those in pressure rise and flow coefficient. This result is similar to that obtained in the baseline cases.

5.3.1 Overall Pressure Rise and Loss

Figures (5.2) and (5.3) show the rotor pressure rise characteristic and loss bucket obtained from steady averaging plane computations, and from time averages of the different unsteady flow simulations. The overall rotor pressure rise and flow coefficients were adequately calculated using the steady flow approximation. As discussed in Chapter 4, this result appears to be fortuitous and should not be expected to hold in general.

The time averaged relative total pressure loss coefficients do not agree with the values obtained from the steady flow computations. As evidenced by the elevated level obtained in the 50% axial gap case, the increase in time averaged rotor loss also appears dependent upon the level of unsteady forcing. To establish this dependence, the rotor mass averaged entropy rise (normalized by that in the axisymmetric exit condition case) is shown in Figure (5.4). The minimum, maximum, and time averaged values are given as a function of the unsteady forcing ($P'/Q_{exit}$) at the mid-axial gap location. This shows that at the peak pressure rise operating condition, the time averaged rotor loss increases with the level of exit unsteadiness. The determination of which regions of the rotor flow field are responsible for the predicted difference in loss will be examined next.
5.3.2 Effect of Downstream Stator Radial Design

The baseline stage configuration described in Chapter 2, and analyzed in Chapters 3 and 4, consisted of a rotor followed by a three-dimensional bowed stator. Bowing the stator reduced loading in the stator endwall regions and increased blade loading away from the endwalls (e.g. midspan). This decreases endwall and corner separations and which benefits the stator performance as shown in Figure (2.5) [55]. The radial variation in stator loading also impacts the upstream rotor two ways. Because the bowed stator endwalls are more lightly loaded, they provide less back pressure variation to the upstream rotor endwall region. Also, for a fixed stage pressure rise, unloading the stator endwalls causes the rotor endwall regions to become more highly loaded. (This can be thought of as changing the degree of reaction for a particular spanwise location.) This second effect has been examined by Graf and Sharma [21] for steady flow in a rotor subjected to different radial back pressure profiles. They showed that compared with straight stators, the use of bowed design results in approximately a 10% increase in rotor endwall loss and blockage for the same average stage pressure rise and flow.

To determine the impact of the downstream stator radial design in the unsteady flow environment, a simulation was conducted with the baseline stator replaced by a straight stator with the same airfoil sections. Figure (5.5) shows the bowed and re-stacked straight stators. The computational grid utilized for the rotor-straight stator simulation had the same dimensions, cell distribution, and overall topology as in the original rotor-bowed stator cases described in Section (2.4.1).

Figures (5.2), (5.3) and (5.4) show that the time averaged rotor pressure rise, flow capacity, and loss were similar with the baseline bowed and nominal straight stator designs. As shown by Graf and Sharma [21], however, the radial distribution of rotor pressure rise and loss is affected by the difference in stator design.

Figure (5.6) shows the time averaged, pitch averaged, rotor loss distribution for the two configurations. The shape of the loss profile in the outer ~20% span is similar for both
configurations due to the unsteady tip vortex motion. However, the level of loss in the tip region is consistently lower with the straight stator, while that along the remainder of the blade is slightly higher. This spanwise re-distribution is consistent with the rotor radial loading profile shown in Figure (5.7). The straight stator design unloads the rotor endwalls slightly; the opposite is true away from the endwalls, and hence the rotor midspan pressure rise and loss are increased.

*Tip Region Loss*

The time averaged rotor tip region entropy rise (defined in Section (3.2.3)) is plotted in Figure (5.8) as a function of the time averaged rotor blade tip loading. All of the bowed and straight cases at the peak pressure operating point are shown with the entropy normalized by that in the axisymmetric exit condition case. From this it appears that for a given rotor design the tip region loss depends on the local pressure rise. This is not surprising since reducing the tip loading is known to decrease the tip leakage mass flux, which in turn affects the loss as shown in Section (4.3.3). In addition, lowering the rotor tip loading lessens the diffusion of the endwall flow thus allowing any endwall mixing processes to occur at a lower absolute pressure level which also decreases the loss.

Figure (5.9) illustrates the dependence of the tip entropy rise on rotor tip region forcing as given by \( P'/Q_{cort} \). Compared to the rotor-bowed stator cases, the rotor-straight stator had lower tip losses at forcing levels both above and below the nominal bowed stator case. This indicates that it is not only the level of unsteady pressure variation which affects the rotor tip loss, but also the time mean tip loading. Despite the differences in time averaged rotor tip loading, however, the time averaged endwall loss was always below that obtained in the steady flow approximations.
Endwall Blockage

The relation between time averaged rotor endwall blockage and tip region ideal loading parameter (defined in Section (3.2.4)) is shown in Figure (5.10). Figure (5.10) includes all of the unsteady cases in the sensitivity study, the steady axisymmetric exit condition case at peak loading, and the experimental data of Khalsa [70]. For all of the unsteady flow cases considered the predicted endwall blockage was generally below that obtained in the steady flow computations. The reasons for this are the same as those given in Section (4.3.4) with regard to the baseline unsteady flow simulations.

For the straight stator cases, Figure (5.10) shows that the blockage and tip loading parameter are decreased compared to the bowed stator results. The reduction in blockage with the straight stator was primarily due to the decrease in rotor endwall loading, while the corresponding change in the tip region ideal loading parameter indicates both lower endwall total pressure loss and loading. This behavior is consistent with the trends obtained by Graf and Sharma [21].

5.3.3 Effect of Rotor-Stator Axial Spacing

To determine the effect of axial spacing on rotor performance, the rotor-straight stator case was utilized as the baseline, and a new case was constructed by reducing the axial gap to 50% of the original. Table [5.5] lists the geometric parameters for the rotor and stator which differ from the baseline rotor-bowed stator configuration given in Tables [2.1] and [2.2].

Table [5.5]: Rotor and Straight Stator Parameters at 50% Axial Spacing

| Axial gap/rotor chord | 0.16 |
| Stator stacking       | Radial |
All other parameters were kept the same as in the baseline cases, and the simulation was conducted at the peak pressure rise operating point.\textsuperscript{21}

The effect of axial spacing on overall rotor pressure rise and flow coefficient is shown in Figure (5.2). The time averaged pressure rise and flow are close to the steady flow approximation, however, as shown in Table (5.3) the level of unsteady fluctuation was larger than in any other case. In addition, as shown in Figures (5.3) and (5.4), the overall rotor loss was greater than in other unsteady flow simulations. To determine which region of the flow field was responsible for the increased loss, the time averaged rotor spanwise loss distributions are shown in Figure (5.11) for the nominal and 50\% spacing cases. Reducing the axial spacing yielded higher loss compared to the nominal case along the inner \textasciitilde80\% of span and lower loss in the tip region. Furthermore, with the 50\% axial spacing, the blade loss and the tip clearance related loss were the highest and lowest respectively, of any examined in this investigation.

\textit{Tip Region Loss}

The shape of the loss profile in Figure (5.11) suggests the presence of tip vortex motion as in the baseline case. Relative to a nominal gap simulation, the increased circumferential back pressure variation increased the tip loss fluctuation as shown in Figure (5.9), although the time averaged endwall loss was slightly below that obtained with nominal spacing. Compared to the baseline bowed stator case, the use of the straight stator reduced the time averaged rotor tip loading and endwall loss as illustrated in Figure (5.8). Therefore, as compared with the nominal spacing results, the reduced rotor-stator spacing slightly decreased the predicted time averaged endwall loss.

\textsuperscript{21} The reduced axial spacing required that the computational grid be modified so as to reduce the number of planes in the rotor-stator gap from 42 to 32. The remaining grid dimensions are as given in Section (2.4.1).
Endwall Blockage

The slight decrease in tip region loss with tight axial spacing translates into further decreases in time averaged endwall blockage. Figure (5.10) illustrates this since both the blockage and tip loading parameter were below those obtained in the other unsteady flow simulations.

5.3.4 Effect of Rotor-Stator Blade Count Ratio

Although the baseline stage utilized in this investigation had a 1:1 blade count ratio, this is generally not representative of the ratios employed in modern multi-stage compressor design. To determine the effects of blade count ratio, a case with 2 rotors and 3 bowed stators was considered, giving a blade count ratio of 1:1.5.\textsuperscript{22}

To achieve the 2:3 ratio, the stator was scaled by holding solidity and airfoil thickness and changing blade aspect ratio. For the baseline bowed stator design given in Table [2.2], the new aspect ratio was 2.07. No modifications to the baseline computational grid were required for this simulation. However, for each blade row the passage periodic boundary conditions were modified so that the multiple rotors and stators would communicate with the appropriate tangentially neighboring blades. As shown in Figure (5.1), to obtain the periodic response associated with the tip vortex required that time averages be performed over 3 stator passing periods. Due to the computational storage requirements of this study, the time averaged rotor performance was computed using only 3 time frames per stator passing (9 total). The individual rotor passages were then averaged together to give the final temporally averaged flow field.

\textsuperscript{22} The design blade count ratio for this stage was 1:1.26 which is approximately half way between those examined.
Overall Rotor Performance

As shown in Table [5.1], the primary effect attributable to the increased stator count was a decrease in amplitude of the circumferential back pressure variation due to the reduction in stator blade pitch as shown in equation (4.2). The time averaged rotor pressure rise and flow were not greatly affected by the blade count ratio as shown in Figure (5.2), and the overall rotor loss was close to the steady flow approximation as illustrated in Figures (5.3) and (5.4). The implication drawn is that a reduction in the imposed back pressure non-uniformity makes the time averaged rotor flow field more like that obtained using circumferential averaging techniques. This trend also held with regard to the spanwise distribution of rotor relative total pressure loss shown in Figure (5.12). Compared to the 1:1 case, the inner ~80% of span had less loss, while the tip region had slightly higher loss.

Tip Region Loss

For the 2:3 blade count case, the period of the tip vortex motion was still close to the rotor flow through time as shown in Figure (5.1), but the vortex motion was reduced as compared to the 1:1 case. In approximate terms, for the 1:1 cases the vortex trajectory moved roughly half a rotor pitch during a single stator passing period (Figure (4.4)), while for the 2:3 case the vortex moved only one-third of a rotor pitch. Increasing the frequency and decreasing the amplitude of the exit pressure variation reduced the oscillation in tip region loss. (In the high frequency-high exit unsteadiness limit, the vortex motion should go to zero.) As a result of the diminished vortex motion, the time averaged tip region loss shown in Figures (5.8), (5.9), and (5.12), was closer to that obtained using steady flow models. As shown in Figure (5.4), the time averaged overall rotor loss was also closer to that computed assuming steady flow, and was lower than that obtained in the other unsteady flow computations.
Endwall Blockage

The endwall blockage, shown in Figure (5.10), is also closer to that in the steady flow prediction. As illustrated in Figure (5.8), since the rotor tip loading was approximately the same as in the 1:1 case, the decrease in unsteady forcing appears to be primarily responsible for the increased blockage. Compared to the baseline unsteady flow simulation, increasing the stator blade count at fixed solidity appears detrimental to the rotor endwall flow field.

5.4 Rotor Response to Circumferentially Specified Exit Condition

In all of the unsteady flow simulations of the stage, the downstream stator represented a spatially and temporally varying pressure boundary condition to which the rotor was subjected. The stator loading oscillated as a result of interaction with the incoming rotor wakes and tip vortices, and hence the pressure field at a given location upstream of the stator was not constant in time. To determine the influence of temporal changes in the exit pressure field, one can consider a rotor subjected to the time averaged stator back pressure profile. In the stator frame of reference the pressure pattern is an approximately sinusoidal circumferential variation with spatial frequency equal to the stator pitch. In the rotor frame, this variation is a moving periodic pressure boundary condition. The unsteady interaction of the rotor with such an exit condition constitutes a unilateral (one-way) interaction.

To execute an unsteady flow computation which simulates the situation described above, the time averaged pressure field from the baseline rotor-bowed stator configuration (the circumferential and radial pressure distribution at the mid axial gap location) was extracted and mapped onto an \((r-\theta)\) grid plane. The grid plane was then positioned behind the rotor at mid axial gap, replacing the stator in the unsteady 2-row simulation. The dimensions of the plane were \(1\times33\times73\), where the axial extent was a single cell while the tangential and radial dimensions were the same as the baseline stator mesh. A separate
plane was utilized since it remained in the stator frame of reference allowing the rotor to move past it. Periodic flow boundary conditions were enforced at the tangential ends of the plane, similar to those used for the blade rows. Because of the reduction in overall grid cell count (only 274,626 cells in total), the computation was executed using only a single workstation.

**Overall Rotor Performance**

Figure (5.2) shows that the time averaged rotor pressure rise and flow were not greatly altered by the circumferentially specified exit condition compared with the other unsteady cases. As shown in Figure (5.3) the overall rotor loss was also close to that from the baseline rotor-bowed stator configuration.

Figure (5.13) shows the time averaged and instantaneous spanwise loss distributions for the rotor with the time averaged stator pressure field and for the baseline stage simulation. The agreement between the profiles in Figure (5.13a) indicates that the time averaged effects of the downstream unsteadiness were largely obtained using the specified exit condition. Figure (5.13b) shows that instantaneous flow behavior was also similar to that in the full stage calculation (Figure (4.14c)). The motion of the high loss tip vortex fluid is evident as is the oscillation in blade profile loss. The period of the vortex motion was given in Figure (5.1) and was close to that obtained in the stage simulation. The conclusion is that the essential fluid dynamic phenomena depend primarily on the time averaged circumferential pressure variation imposed by the stator, and not on instantaneous changes brought about by interaction with the upstream rotor.

**5.5 Behavior of an Isolated Rotor**

Throughout the investigations described the rotor has always been subjected to continuous unsteady forcing from a downstream pressure non-uniformity. Despite differences in amplitude, frequency, and radial content of imposed back pressure
variations, the qualitative response of the rotor flow field was similar including motion of the tip vortex, interaction with the wake of the adjacent rotor blade, and agreement between the period of the vortex motion and the rotor flow through time. Because of this, it was hypothesized that the vortex motion might be self-induced.

The existence of self-induced vortex motion in turbomachine blade passages has previously been experimentally observed for secondary flow vortices in turbines by Sharma et al [78]. They found quasi-periodic motion of these vortices to occur in an isolated cascade of turbine nozzle guide vanes.

To determine if the tip vortex motion identified in this study might be a self-excited phenomenon, a time accurate simulation of an isolated rotor was performed. Two separate goals were set for this computation. The first was to identify whether a fluid dynamic instability or self-induced oscillation was responsible for the vortex motion, and the second was to establish if the numerical methods and/or models employed in the NiSTAR solver could have potentially given rise to the observed behavior.

The study utilized the same geometry, computational grid, and boundary conditions as those described in Section (4.3.2) for the steady flow modeling of an isolated rotor subjected to an axisymmetric exit pressure profile. The time accurate calculation of the rotor was initiated from this steady flow field at the peak pressure rise operating condition. The time step and the numerical method/approximations were the same as those utilized in baseline rotor-bowed stator unsteady flow studies.

**Rotor Flow Response**

Initially, the rotor flow field was simulated for 3 rotor flow through times to determine if self-excited motion of the tip vortex would occur. This length of time was chosen because, as shown in the previous two-row results, the period of the vortex motion was approximately a single flow through time.
Throughout the simulation, qualitative examination of the rotor exit flow field and time traces of various flow quantities provided no indication of tip vortex motion. However, this did not conclusively demonstrate that the system would not exhibit oscillations if perturbed (i.e. if disturbed from its initial state the vortex may begin to oscillate). A downstream planar pulse increase in rotor back pressure was thus introduced to try and initiate the vortex motion. The pulse increased the back pressure to the same level as the maximum encountered during the unsteady rotor-stator interaction. The duration of the pulse was chosen to be one-quarter of a stator passing period to provide a disturbance similar to the modulation given by the stator row.\textsuperscript{23}

To illustrate the impact of the disturbance on the tip vortex, Figure (5.14) shows static pressure contours in a tangential plane at the center of the rotor tip clearance gap. The first image, Figure (5.14a), was taken after the three initial flow through time periods and appears similar to that obtained in the steady flow approximations shown in Figure (3.5b). No vortex motion was present. The back pressure pulse was then applied, and the second picture, Figure (5.14b), was obtained at the end of the pulse period. Compared to (5.14a), the vortex trajectory is slightly more tangential and the origin of the reduced static pressure associated with the leakage flow has moved closer to the rotor leading edge. This demonstrates that the vortex can move by changing the exit pressure.

Figure (5.15) shows the rotor pressure ratio as a function of flow through time during the simulation. The pressure ratio increased rapidly after the pulse was applied, and then remained nearly constant for one-eighth the rotor flow through time (one-quarter of a stator passing period). Once the exit pressure was returned to the initial value, the pressure ratio exhibited a highly damped oscillation back to the pre-pulse condition. No sustained or self-excited vortex motion was observed. The final recovered post-pulse flow field is

\textsuperscript{23} Based on the propagation speed of upstream traveling disturbances $(V - a)$, this time scale was sufficient for the pressure pulse to reach the rotor trailing edge.
shown in Figure (5.14c) and appears virtually indistinguishable from that prior to the disturbance.

The rotor pitch averaged spanwise loss distribution is also shown in Figure (5.16) along with that in the steady axisymmetric exit condition case. The loss profile is that obtained after recovery from the pressure pulse, and corresponds with the flow field shown in Figure (5.14c). The similarity in shape between the profiles shown in Figure (5.16) (especially in the tip region) indicates that the time accurate and steady flow responses of an isolated rotor are essentially the same. The shape of the loss profile is not like that obtained from full stage simulations which indicated tip vortex motion (e.g. Figure (4.13)).

This computational experiment was not conclusive but it showed that the tip vortex motion observed in the rotor-stator unsteady flow simulations could not be excited by perturbing the rotor with a discrete, short duration (0.25 stator passing period), back pressure disturbance. Also, for the case examined the numerical methods and models utilized were not responsible for the observed response of the tip vortex.

5.6 Conclusions from the Sensitivity Study

Through the use of controlled numerical experiments, a sensitivity study of unsteady rotor-stator interaction has been conducted. The following summarizes the observations and conclusions:

**Observed Flow Features**

- Tip vortex motion was observed at the rotor exit in all rotor-stator simulations.
- The period of the vortex motion scaled with the rotor flow through time, not the stator passing period.
- Tip vortex merged with the wake of the adjacent rotor blade in the rotor exit plane during a portion of the vortex motion cycle. This resulted in oscillations in instantaneous tip region loss and endwall blockage.
- An oscillatory response of the tip vortex could not be excited by a discrete back pressure pulse which increased the rotor exit pressure level to the maximum
encountered during the unsteady rotor-stator interaction and had duration equal to one-quarter of a stator passing period.

**Performance Implications**

- The rotor time averaged pressure rise and flow capacity were close to that predicted assuming steady flow.

- The time averaged overall rotor loss was found to increase with the amplitude of the imposed back pressure variation.

- The time averaged rotor tip region loss and endwall blockage depended on both time mean rotor tip loading and amplitude of the imposed back pressure variation.
  - Lowering the tip loading reduced tip region loss,
  - Increasing the amplitude of the exit unsteadiness imposed on the rotor only slightly reduced the tip loss.

- Changing the downstream stator radial geometry from bowed to straight altered the rotor spanwise loading distribution thus reducing the endwall pressure rise and loss.

- Reducing the rotor-stator axial spacing to 50% of nominal increased the amplitude of the unsteady forcing imposed on the rotor, increased the overall rotor loss, but resulted in a reduction in tip region loss.

- Increasing the stator blade count reduced the amplitude and increased the frequency of the unsteady forcing imposed on the rotor. Both these effects caused the rotor flow field to appear more like that obtained assuming steady flow.

- The effect of unsteady interaction on the rotor was predominantly unilateral (one-way) and could be obtained using a time averaged, circumferentially varying, stator imposed back pressure.
Figure (5.1): Correlation of rotor tip vortex periodicity with rotor flow through time

Figure (5.2): Rotor pressure rise characteristic
Figure (5.3) Rotor mass averaged relative total pressure loss bucket

Figure (5.4): Maximum, time averaged, and minimum mass averaged entropy rise for the rotor
Figure (5.5): Bowed stator and corresponding straight (re-stacked) stator

Figure (5.6): Rotor pitch averaged spanwise loss profiles with bowed and straight downstream stators
Figure (5.7): Rotor spanwise loading distributions with bowed and straight downstream stators

Figure (5.8): Correlation of time averaged entropy rise for the rotor tip region with rotor tip loading
Figure (5.9): Maximum, time averaged, and minimum mass averaged entropy rise for the rotor tip region

Figure (5.10): Rotor tip region flow blockage correlation (data from Khalsa [70])
Figure (5.11) Rotor pitch averaged spanwise loss profiles with the downstream straight stator at nominal axial spacing and at 50% of the nominal axial spacing.

Figure (5.12) Rotor pitch averaged spanwise loss profiles with the downstream bowed stator having (1r:1s) and (2r:3s) blade count ratios.
Figure (5.13): Rotor pitch averaged spanwise loss profiles with the circumferentially specified downstream pressure variation
Figure (5.14): Time accurate response of an isolated rotor to a back pressure pulse

Figure (5.15): Pressure ratio as a function of time for the isolated rotor subjected to a back pressure pulse
Figure (5.16): Rotor pitch averaged spanwise loss profiles from the time accurate and steady axisymmetric B.C. simulations of the isolated rotor.
Chapter 6
Conclusions

6.1 Summary and Conclusions

Three dimensional steady and time accurate Reynolds-averaged Navier-Stokes computations have been utilized to investigate the effects of stator pressure field on upstream rotor performance. The analyses showed that the unsteady response of the rotor tip vortex is a dominant factor in determining the time averaged flow structure in the endwall region.

As a result of unsteady rotor-stator interaction the time averaged rotor loss and endwall blockage were different than those predicted using steady flow models. As compared to an isolated rotor at the same overall pressure rise, the time averaged rotor loss was found to increase by 20 to 40% depending upon the amplitude of the stator imposed back pressure variation. The time averaged loss and flow blockage in the endwall region (outer ~20% span) decreased with reduced rotor tip loading and with increased amplitude of the back pressure non-uniformity. The changes in the rotor endwall flow field resulting from increased exit unsteadiness were associated with the response of the tip vortex in the unsteady flow environment.

In all of the unsteady rotor-stator simulations, radial and tangential motions of the tip vortex on the order of the blade pitch was observed at the rotor exit. The fluid dynamic period of the vortex motion was found to scale with rotor flow through time rather than the stator passing period. Due to the motion the tip vortex periodically merged with the wake of the tangentially adjacent rotor blade resulting in large oscillations in rotor tip region loss
and endwall blockage. The movement of the vortex also enhanced mixing in the endwall region causing the time averaged tip clearance related loss to be within 20% of that computed assuming fully mixed out conditions.

A sensitivity study of unsteady flow effects was also conducted to determine the generality of the baseline results to different geometric and aerodynamic conditions representative of modern high pressure compressor designs.

**Effect of Stator Radial Geometry Distribution**

The stator radial design was changed from bowed to straight which altered the time averaged rotor spanwise loading distribution. Compared to the bowed design, the straight stator reduced the rotor tip loading resulting in a 20 to 30% decrease in time averaged endwall loss and blockage.

**Effect of Rotor-Stator Axial Spacing**

For the rotor-straight stator case, a 50% reduction in blade row axial spacing increased (nearly doubled) the amplitude of the forcing imposed on the rotor. This decreased the rotor endwall loss and blockage to the lowest levels obtained in this study (approximately 30% below the baseline rotor-bowed stator results).

**Effect of Rotor-Stator Blade Count Ratio**

For the baseline rotor-bowed stator case, increasing the stator blade count at fixed solidity reduced the amplitude of the forcing imposed on the rotor. This diminished the effects of unsteadiness and lead to a 10% increase in endwall loss and blockage as compared to the 1 rotor: 1 stator blade count configuration.
Rotor Response to Circumferentially Specified Exit Condition

The unsteady rotor response was found to be predominantly one-way and can be assessed by specifying the time averaged, circumferentially varying, stator back pressure distribution at the rotor exit. In the present case the rotor flow field was shown not to depend strongly upon instantaneous changes of the stator pressure field from the time mean.

Behavior of an Isolated Rotor

A time accurate simulation of an isolated rotor showed that without downstream forcing, the unsteady vortex motion was not self-induced and could not be initiated by a discrete, short duration (= 0.25 stator passing period) pulse in rotor back pressure.

Three steady flow approximations of the rotor-stator interaction problem were also examined to determine if the time averaged effects of unsteadiness on the rotor flow field could be captured using the existing models. The models were: (1) an axisymmetric representation of the stator pressure field, (2) an “industry-standard” inter-blade row averaging plane technique, and (3) a model which incorporated both the stator pressure field and corresponding flow blockage effects (bodyforces and deterministic stresses). For this investigation the steady flow methods did give the time averaged rotor pressure rise and flow capacity. However, the steady flow approximations were generally unable to accurately capture the magnitudes of the time averaged effects of unsteadiness on rotor loss and endwall blockage. The differences ranged from 5 to 50% of the time averaged values depending upon the amplitude of the downstream pressure non-uniformity, the stator design, and the stage operating condition.
6.2 Implications of Present Work

The results of the numerical experiments suggest possible modifications to stage designs to improve performance in the unsteady flow environment.

The present study indicates that lowering the time averaged rotor tip loading can potentially improve pressure rise capability (i.e. stability) and efficiency (i.e. loss generation), because sizable reductions in endwall loss and blockage were achieved with modest changes in blade loading. Additional improvements in the endwall flow field were also obtained by closely coupling the rotor and stator. This suggests that a combination of unloading the rotor tip while closely coupling the blade rows in the tip region would improve the rotor endwall flow field. However, an increase in rotor profile loss and overall stator loss was observed when the blade rows were more closely coupled, and hence a trade off between these effects may be warranted.

The use of rotor-stator blade count ratio as a lever to change the nature of the unsteady blade row interaction also appears possible. Increasing the stator count at fixed solidity decreased the overall effects of unsteadiness on the rotor and produced a flow field which was more like that obtained assuming steady flow in the relative frame. This benefited the rotor profile loss, while increasing tip region loss and the overall stator loss. This suggests than an increase in stator blade count while simultaneously close coupling the rotor tip with the stator may improve both the endwall and profile losses. The effects on the downstream stator performance are however not clear at present.

6.3 Recommendations for Future Work

The following list summarizes some directions in which the present study could be extended or enhanced.

- The flow fields examined were obtained from computational fluid dynamic codes. Detailed unsteady flow measurements from physical experiments should be obtained to validate the performance trends and flow response found here.
- Rotor-stator unsteady interaction is known to be more severe in the high speed (e.g. transonic) flow environment. Computational extensions to this flow regime should be explored, especially with regard to front stages of HPC's and multi-stage fans.

- The fluid dynamic behavior of the stator subjected to unsteady inflow associated with the rotor tip vortex should be examined with regard to the formation of loss in the stator passage at increased levels of inlet flow unsteadiness.

- The unsteady flow behavior of a stator-rotor (as opposed to rotor-stator) stage should be examined to determined how inlet unsteadiness impacts the rotor performance.

- The generation of loss in unsteady blade boundary layers at frequencies characteristic of rotor-stator interaction should be examined since a large fraction (~70%) of the time averaged aerodynamic loss is associated with this.

- The impact of casing treatment on the rotor endwall flow response should be examined through the use of unsteady flow computations.

- Two avenues appear to be of interest with regard to improving the computational capability of current CFD codes.
  
  First, because the time scales associated with the unsteady flow environment are generally not known a priori, running time averages of the flow field are suggested as means to obtain accurate representations of the time averaged aerodynamic phenomena. Such averages could be generated over several flow through times once periodicity of the solution has been established.

  Second, current steady flow models should be modified to incorporate approximations which more accurately represent the time averaged effects of unsteadiness in the endwall region. This may be possible through the inclusion of mixing or loss models which are based on measurements and computations of the unsteady flow environment.
References and Bibliography


[38] Kumar, A., Kerrebrock, J. L., "Rotor wake transport in turbomachine stators," Massachusetts Institute of Technology, Department of Aeronautics and Astronautics, Gas Turbine Laboratory Report No. 103.


Appendix A

Summary of CFD Calculations Executed

The computational fluid dynamic simulations listed here were conducted using the NiSTAR and NASTAR codes. This information provides the average operating conditions for the stage or rotor obtained from the study. Inlet profiles and baseline geometry for all of the computations are presented in Chapter 2. All pressures listed in the following tables are area-averaged while flow coefficient is density-averaged.

A.1 Steady Flow

NiSTAR Simulations

<table>
<thead>
<tr>
<th>Case</th>
<th>$\Delta P/Q_{in}$</th>
<th>$P_{out}/P_{in}$</th>
<th>$\phi$</th>
<th>$\dot{m}_{cor}$ (lb/s)</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>rs13yf</td>
<td>0.5370</td>
<td>1.162</td>
<td>0.4818</td>
<td>22.19</td>
<td>stage</td>
</tr>
<tr>
<td>rs13ym</td>
<td>0.5255</td>
<td>1.160</td>
<td>0.4921</td>
<td>22.62</td>
<td>stage</td>
</tr>
<tr>
<td>rs13ya</td>
<td>0.5150</td>
<td>1.158</td>
<td>0.4968</td>
<td>22.84</td>
<td>stage</td>
</tr>
<tr>
<td>rs13yn</td>
<td>0.4964</td>
<td>1.153</td>
<td>0.5040</td>
<td>23.15</td>
<td>stage</td>
</tr>
<tr>
<td>rs13yh</td>
<td>0.4822</td>
<td>1.149</td>
<td>0.5099</td>
<td>23.42</td>
<td>stage</td>
</tr>
<tr>
<td>rs13yj</td>
<td>0.4466</td>
<td>1.139</td>
<td>0.5144</td>
<td>23.61</td>
<td>stage</td>
</tr>
<tr>
<td>rs13ykb</td>
<td>0.5760</td>
<td>1.175</td>
<td>0.4858</td>
<td>22.41</td>
<td>stage, dense tip grid</td>
</tr>
<tr>
<td>rs13yka</td>
<td>0.5447</td>
<td>1.66</td>
<td>0.4963</td>
<td>22.85</td>
<td>stage, dense tip grid</td>
</tr>
<tr>
<td>rs13exh</td>
<td>0.3758</td>
<td>1.114</td>
<td>0.4810</td>
<td>22.27</td>
<td>rotor alone axisymmetric exit B.C.</td>
</tr>
<tr>
<td>r13exhl</td>
<td>0.3480</td>
<td>1.108</td>
<td>0.5108</td>
<td>23.44</td>
<td>rotor alone axisymmetric exit B.C.</td>
</tr>
</tbody>
</table>
NASTAR Simulations

<table>
<thead>
<tr>
<th>Case</th>
<th>$\Delta P/Q_{in}$</th>
<th>$P_{exit}/P_{in}$</th>
<th>$\phi$</th>
<th>$\dot{m}_{cor}$ (lb/s)</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>rs13nd</td>
<td>0.6290</td>
<td>1.188</td>
<td>0.4687</td>
<td>21.48</td>
<td>stage</td>
</tr>
<tr>
<td>rs13nb</td>
<td>0.6150</td>
<td>1.185</td>
<td>0.4750</td>
<td>21.82</td>
<td>stage</td>
</tr>
<tr>
<td>rs13na</td>
<td>0.5730</td>
<td>1.173</td>
<td>0.4871</td>
<td>22.58</td>
<td>stage</td>
</tr>
<tr>
<td>rs13nc</td>
<td>0.5360</td>
<td>1.164</td>
<td>0.4994</td>
<td>23.04</td>
<td>stage</td>
</tr>
<tr>
<td>rs13ne</td>
<td>0.4850</td>
<td>1.149</td>
<td>0.5090</td>
<td>23.43</td>
<td>stage</td>
</tr>
<tr>
<td>rs13ng</td>
<td>0.4580</td>
<td>1.142</td>
<td>0.5150</td>
<td>23.67</td>
<td>stage</td>
</tr>
</tbody>
</table>

A.2 Unsteady Flow

NiSTAR was used exclusively for all unsteady flow simulations. Time averaged values for the stage or rotor are listed below.

<table>
<thead>
<tr>
<th>Case</th>
<th>$\Delta P/Q_{in}$</th>
<th>$P_{exit}/P_{in}$</th>
<th>$\phi$</th>
<th>$\dot{m}_{cor}$ (lb/s)</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>uns</td>
<td>0.5336</td>
<td>1.1611</td>
<td>0.4804</td>
<td>22.15</td>
<td>stage, 1R-1S, nom. axial gap</td>
</tr>
<tr>
<td>unslow</td>
<td>0.4810</td>
<td>1.1484</td>
<td>0.5060</td>
<td>23.18</td>
<td>stage, 1R-1S, nom. axial gap</td>
</tr>
<tr>
<td>uns23</td>
<td>0.5334</td>
<td>1.1611</td>
<td>0.4808</td>
<td>22.12</td>
<td>stage, 2R-3S, nom. axial gap</td>
</tr>
<tr>
<td>bow</td>
<td>0.3781</td>
<td>1.1150</td>
<td>0.4800</td>
<td>22.11</td>
<td>rotor alone with circ. specified B.C.</td>
</tr>
<tr>
<td>strte</td>
<td>0.5313</td>
<td>1.1609</td>
<td>0.4811</td>
<td>22.22</td>
<td>stage, 1R-1S, straight stator, nom. axial gap</td>
</tr>
<tr>
<td>strtd</td>
<td>0.5391</td>
<td>1.1633</td>
<td>0.4820</td>
<td>22.30</td>
<td>stage, 1R-1S, straight stator, 50% axial gap</td>
</tr>
<tr>
<td>r13tim</td>
<td>0.3787</td>
<td>1.1149</td>
<td>0.4845</td>
<td>22.31</td>
<td>rotor alone, axisymmetric B.C., time accurate</td>
</tr>
</tbody>
</table>

153
Appendix B

Stator Unsteady Flow Behavior

B.1 Introduction

In this Appendix the results from the unsteady flow simulations are examined with regard to the stator flow field. Because the primary goal of this investigation was to characterize the rotor flow field in the unsteady flow environment, details of the flow within the stator were not focused upon.

The results from the baseline rotor-bowed stator cases at high and low loading will be given first. The bowed stator flow field and its response will be contrasted to that obtained in the steady flow computations using the averaging plane technique. Finally, results from the sensitivity study are presented along with conclusions.

B.2 Fluid Dynamic Features

Steady flow models of the multi-row compressor environment utilize averaging techniques to allow flow information to be passed from rotating to stationary blade rows and vice versa. Because of this the inlet conditions to a particular blade row are often modeled as having spatial variations which are invariant in time. For three-dimensional flows the most common method is circumferential averaging of the flow entering a particular row. An example of such inlet conditions can be seen in Figure (2.8). The success of such approximations in accurately representing the actual flow environment is dependent upon the degree of inlet flow non-uniformity and unsteadiness. As shown in
Chapter 4 the rotor wake and tip vortex are the main forms of periodic unsteadiness which the downstream stator encounters. Therefore these will be discussed below.

*Wake Interaction*

The chopping of rotor wakes and subsequent flow migration in the stator blade passages can be observed in Figure (4.3). The cutting of the wakes is a kinematic consequence arising from the relative motion of the blade rows; the wake migration towards the pressure surface of the adjacent stator can be explained by considering kinematics. As shown by Kerrebrock and Mikolajczak [40] the difference in rotor relative velocity between rotor wake and core flow results in the creation of slip velocity in the absolute frame. The slip velocity vector is directed towards the stator pressure surface and is responsible for the wake migration and the total temperature segregation experimentally observed in stator blade passages.

*Tip Vortex Interaction*

The interaction of the tip vortex with the adjacent stator is shown in Figure (B.1). This figure shows entropy contours at 97% span at 4 time instants during one stator passing period. The rotor tip vortex appears as a region of high entropy fluid which has periodically entered the stator passage and migrated towards the pressure surface as it convects through. The tangential motion of the vortex observed at the rotor-stator mid axial gap in Figure (4.4) is associated with movement of this high entropy region. The mechanism responsible for the cross-passage migration appears similar to the slip velocity for the wake region. The vortex region is effectively a velocity defect and so its slip velocity is analogous to that of the wake. The periodicity of the vortex motion was identified in Chapter 4 as being approximately $1.7 \pm 0.1$ stator passing periods. Although only a single stator passing period is shown in Figure (B.1), the entropy disturbance created by the vortex at the stator inlet was found to be periodic with this frequency. Based
on this, the time averaged stator performance was obtained by averaging the flow over two-stator passing periods in the same manner as that for the rotor.

B.3 Stator Performance Characteristics

The impact of the incoming unsteady flow on stator performance is provided for the baseline cases at high and low loading.

B.3.1 Pressure Rise Capability

The stator flow field response in the steady flow environment was given in Section (3.3). There it was shown that the stator remained unseparated over the range of flow coefficient examined in this investigation. Figure (B.2) illustrates the steady flow characteristic along with the locus of operation found from the unsteady flow simulations. The time averaged stator pressure rise is within 2% of that in the steady flow analysis. The blade row time averaged pressure rise is thus essentially determined by the average inlet and exit conditions. The level of pressure rise fluctuation at high loading is ±6% of mean and decreases as the stator/stage loading is reduced.

Figure (B.3) shows the pitch averaged spanwise stator loading distribution at twenty time instants during two blade passing periods at the high loading condition. For the inner 75% of span the stator loading varies by no more than ±5% of the mean. The outer 25% has maximum oscillations on the order of ±15% of the mean. The stator exit pressure was specified as uniform one-span downstream, and thus the fluctuations in loading are associated with changes in stator inlet conditions.

Figure (B.4) shows the pitch averaged inlet flow angle obtained at the midspan and at 90% span. The periodic entry of the rotor wake at midspan is evident while the disturbance associated with the incoming tip vortex is visible at 90% span. The stator provides a mean turning of approximately 35°, hence the maximum variation in midspan inflow angle is (assuming inviscid incompressible flow) equivalent to ±4.5% of midspan
loading. (The stator loading is defined as the static pressure rise normalized by the inlet
dynamic pressure.) Similarly, the variation in the tip region is equal to ±13% in tip
loading. These values agree well with the loading oscillations found in the unsteady
computation. Although these effects are present at reduced loading the magnitude of the
flow non-uniformities entering the stator are considerably smaller (±1° in the tip inlet angle)
and the level of loading variation at low loading is also attenuated. Thus changes in inflow
angle, and inlet not dynamic pressure, are primarily responsible for the unsteady
modulation in stator pressure rise.

B.3.2 Aerodynamic Losses

Since the flow entering the stator is unsteady, the flow exiting the blade row will
also vary in time. If time traces of stator inlet and exit mass averaged total pressure are
examined a phase relationship is observed. If there is a peak in inlet total pressure at time
\( t \), then the exit total pressure will be found to peak at \( t + t_f \), where \( t_f \) is approximately
the stator flow through time. Therefore, the instantaneous and time averaged loss
coefficients have been computed by taking differences which account for the finite flow
through time. For the time averaged loss this,

\[
\overline{\omega'} = \frac{1}{\Delta T} \int_T^{T+\Delta T} \frac{P_{T1}(t) - P_{T2}(t + t_f)}{P_{T1}(t) - P_1(t)} \, dt
\]  

(B.1)

where

\[
t_f = \frac{b_f}{V_s}
\]  

(B.2)

The coefficients obtained in the manner will be denoted as being phase corrected. This
approach was also utilized for computations of the stator entropy rise coefficient.

Unsteady Loss Variation

Figure (B.4) shows the loss bucket from the steady averaging plane computations
along with the phase corrected instantaneous and time averaged values. There is an oscillation in loss coefficient and flow angle at both high and low loading, but the difference in time average loss is only appreciable (≈30%) at high loading.

Figure (B.6) shows the phase corrected, mass averaged entropy rise variation obtained for the stator at high loading. The tip region (outer ~10% span) shows the periodic behavior associated with the tip vortex and rotor wake. The peak at ~50% blade passing is linked to the wake while that at ~160% is related to the vortex. The periodicity of this trace also coincides with the vortex period identified in Chapter 4. In contrast, the midspan region shows peaks at ~50% and ~150% corresponding to the rotor wakes. The magnitude of entropy fluctuation in the tip region is large with the maximum instantaneous value being nearly twice the mean. Comparing this with the midspan time trace, the tip region appears primarily responsible for the fluctuations in loss shown in Figure (B.5).

Figure (B.7) shows the phase corrected, mass averaged, entropy rise plotted against the stator pressure rise fluctuation. The minimum, maximum, and time averaged values are given, and the entropy rise is normalized by that obtained from the steady averaging plane simulations at the same mean loading. This result shows that the minimum, maximum, and time averaged loss all increase with the level of loading variation.

B.4 Discussion of Sensitivity Study

The overall stator loss behavior in each of the cases run for the sensitivity study conducted of Chapter 5 was examined and this will be briefly addressed in this section.

Effect of Stator Radial Design

The stage flow field was computed at high loading using both bowed and re-stacked straight stators. The flow range over which the stage operated was chosen such that even at maximum loading, the steady flow within the stator row remained unseparated.
For the baseline bowed and straight designs the overall loss response was similar as illustrated in Figure (B.7).

*Effect of Rotor-Stator Axial Spacing*

For the straight stator at 50% of the nominal axial gap, the qualitative behavior was similar to that observed in the baseline cases. The unsteady stator loading fluctuation was however greater, and Figure (B.7) indicates the resultant increase in entropy rise.

*Effect of Rotor-Stator Blade Count Ratio*

The stator loss variation computed for the 2 rotor: 3 stator blade count case was the ensemble average for the three passages. As shown in Chapter 5, the reduced stator size which accompanied the increased blade count produced less peak-to-peak back pressure variation on the upstream rotor. As a result, the magnitudes of the velocity non-uniformity associated with the rotor wake and tip vortex were found to be greater than in the 1:1 blade count ratio cases. This caused the stator to experience larger loading and loss oscillations as shown in Figure (B.7), which decreased the time averaged stator performance.

**B.5 Summary**

- The stator unsteady and time averaged loss was found to increase with the level of loading fluctuation.
- The stator overall time averaged losses were above those obtained in steady flow.
- The stator entropy rise and inlet flow angle variations show periodicity with the observed rotor flow structures.
- The stator unsteady loading was primarily influenced by the variation of inlet flow angle (analogous to gust response).

It should again be mentioned that a detailed examination of the stator flow field was not conducted and it is recommended that further studies be performed to isolate the fluid dynamic mechanisms responsible for the observed behavior.

159
Figure (B.1): Entropy contours at 97% span at 4 instants of time showing interaction of tip clearance vortex with the downstream stator
(contour increments = 20% of stage time averaged entropy rise)
Figure (B.1): - Continued. (contour increments = 20% of stage time averaged entropy rise)
Figure (B.2): Stator pressure rise characteristic

Figure (B.3): Stator instantaneous spanwise loading distribution at peak pressure rise
Figure (B.4): Stator inlet flow angle variation

Figure (B.5): Stator mass averaged total pressure loss bucket
Figure (B.6): Stator phase corrected mass averaged entropy rise variation

Figure (B.7): Maximum, time averaged, and minimum phase corrected mass averaged entropy rise