STRUCTURED AND UNSTRUCTURED COMPUTATIONS OF UNSTEADY TURBOMACHINERY FLOWS USING PRESSURE BASED METHODS

A Thesis in
Aerospace Engineering

by

Wen-Sheng Yu

© 2001 Wen-Sheng Yu

Submitted in Partial Fulfillment of the Requirements for the Degree of

Doctor of Philosophy

December 2001
We approve the thesis of Wen-Sheng Yu.

_____________________                                                             ______________
Robert F. Kunz
Adjunct Assistant Professor of Aerospace Engineering
Thesis Co-Advisor
Co-Chair of Committee

_____________________                                                             ______________
Philip J. Morris
Boeing/A.D. Welliver Professor of Aerospace Engineering
Thesis Co-Advisor
Co-Chair of Committee

_____________________                                                             ______________
Lyle N. Long
Professor of Aerospace Engineering

_____________________                                                             ______________
Vigor Yang
Professor of Mechanical Engineering

_____________________                                                             ______________
Dennis K. McLaughlin
Professor of Aerospace Engineering
Head of the Department of Aerospace Engineering

Date of Signature
ABSTRACT

This thesis is aimed at an improved understanding of unsteady turbomachinery flow physics using time-accurate computational fluid dynamics (CFD) techniques. Two pressure-based algorithms have been adapted for the numerical computation of turbulent, two- and three-dimensional, steady and unsteady flows through turbomachinery. The differential model employed is the incompressible Reynolds averaged Navier-Stokes equations. In this work, several new and/or modern techniques have been adapted, implemented, and applied. These include higher order accurate structured and hybrid unstructured discretization, an inlet wake passing strategy, an interface sliding technique for the computation of rotor-stator interactions, parallel processing capability, and full-two-fluid modeling for multiphase flow analysis. The methods are employed in the analysis of several unsteady turbomachinery flows. The results are presented, compared with experimental data and interpreted, elucidating several important intrinsically unsteady physical processes in these machines.

Details of the physical and numerical modeling strategies of these algorithms are presented, with emphasis placed on new contributions. Specifically, details of the wake passing and rotor-stator interaction schemes are presented, as are details of the discretization practices, structured and unstructured grid generation strategies, multiphase flow analysis treatments, and parallel processing implementation.

Several unsteady turbomachinery test cases are computed and compared with available experimental data. These results illustrate the effectiveness and generality of the
schemes developed, and elucidate important, unsteady physical processes in the machines considered.

First, the unsteady flow field through the midspan section of a second stage stator of a two-stage compressor is carried out with the inlet wake passing strategy. The effects of rotor-stator blade row spacing and the rotor/stator blade count ratio on the turbomachinery unsteady flows are investigated. It is found that a decrease in the rotor-stator blade row spacing causes more decay of the upstream wake and leads to increases in the unsteady pressure fluctuations on the blade surface. The nonlinear effects and variations in the harmonic content are captured by the Navier-Stokes code. Loss mechanisms are evaluated. They include: (1) losses due to the decay of the wake upstream of the blade row; (2) losses due to the decay of the wake inside the stator passage, including modification of the wake profile by the stator pressure field; and (3) the blade profile losses, which are affected by the passing of the inlet wake. The wake.blade count ratio effect is found to be dominated by the reduced frequency effect near the stator leading edge. The wake interaction and vortex formation have major influence on the unsteady pressure response in the 10-25% axial chord region. Beyond this region, the smallest wake.blade count ratio has the most pronounced influence on the unsteady flow and pressure field. The increased interaction effect at higher wake.blade count ratio results in a more rapid decay of the wake, thus reducing the influence of the wake beyond the 25% chord location. The losses increase significantly as the wake.blade count ratio is increased. This increase is mainly brought about by the increased number of wakes and the losses associated with their decay upstream and inside the blade passage.
Second, the three-dimensional, steady state flow field through the inlet guide vane of a high Reynolds number pump is predicted and compared with experimental data. Separations on the blade suction surface near the trailing edge and at the hub wall corner are captured, though the predicted separation zones are smaller that those measured. Loss profiles including profiles losses and wake losses are analyzed. Details of the predicted wake profiles at various radial locations are compared with measurement. Radial and tangential components of the wake profiles are captured well, but an overprediction of the axial wake depth is found at most radial locations. This may be due to the grid and the low Reynolds number two-equation turbulence model used in the computation. Also, it is found from a simulation study at the midspan section that the inlet turbulence length scale can have a significant effect on the decay characteristic of the wakes. Finally, analysis of the decay of the wake and the secondary flow shows that near the endwalls, the decay of the secondary flow is slower than the decay of the wake. Thus the secondary flow could be a major source of unsteadiness for the downstream blade row in these regions.

Third, the midspan section of an axial flow turbine stage is investigated. The nozzle is closely followed by a downstream rotor. The small axial gap between the blade rows indicates a strong unsteady interaction between them. Here, the full rotor-stator interaction is captured by computing the nozzle and rotor flows together simultaneously using the interface sliding scheme, which allows accurate and efficient resolution of the nozzle wake and its interaction with the rotor flow field. The predicted nozzle wake decay is smaller than that measured, likely due to the two-equation turbulence model used. This leads to a higher predicted unsteady response on the rotor blade than that measured. Tur-
bulence intensity, total relative velocity and its unsteady fluctuations are used as indicators to investigate the dynamics of the nozzle wake inside and downstream of the rotor blade passage. The chopping by the rotor blade, the rapid transport of the nozzle wake along the rotor suction surface, and its interaction with the rotor wakes of the nozzle wakes are captured with reasonable accuracy.

Lastly, single- and multi-phase (i.e., bubbly) flows through the midspan section of an axial flow pump stage are investigated. This flow field is of relevance to marine propulsion applications; in particular, the bubbly flow calculation is of interest in surface ship wake acoustic signature analysis. The hybrid unstructured, full rotor-stator, multi-bubble-field (separate continuity and momentum equations solved for five gas “fields” each with a unique representative bubble diameter) analyses performed represents the first demonstration of such a calculation capability that has appeared.
# TABLE OF CONTENTS

LIST OF FIGURES ............................................................................................................. x

LIST OF TABLES ............................................................................................................... xvi

NOMENCLATURE .......................................................................................................... xvii

ACKNOWLEDGEMENTS ............................................................................................... xxii

CHAPTER 1. INTRODUCTION.......................................................................................... 1
  1.1. Background and Justification ............................................................................... 1
  1.2. Objectives and Scope ......................................................................................... 6
  1.3. Literature Review ............................................................................................... 10
    1.3.1. Linearized Analysis of Unsteady Flows .................................................. 10
    1.3.2. Passage-Averaged Aerodynamic Analysis ............................................. 11
    1.3.3. Time Accurate Analysis for Rotor-Stator Interaction ............................ 13
    1.3.4. Grids Used in Turbomachinery Flow Computations ......................... 18
    1.3.5. Multiphase Flows; Parallel Computing ................................................. 20
  1.4. Outline of Thesis ............................................................................................... 21

CHAPTER 2. GOVERNING EQUATIONS AND NUMERICAL PROCEDURES ............ 23
  2.1. The Governing Equations ............................................................................... 23
  2.2. Data Structure .................................................................................................. 26
  2.3. Discretization Schemes .................................................................................... 28
    2.3.1. Discretization Schemes for General Transport Equations in Code A .... 28
    2.3.2. Discretization Schemes for General Transport Equations in Code B .... 32
    2.3.3. Pressure Equation for Code A ............................................................... 35
    2.3.4. Pressure Equation for Code B ............................................................... 40
  2.4. Boundary Conditions ....................................................................................... 41
  2.5. Solution Procedure for Code B ....................................................................... 44
    2.5.1. Transport Scalars ................................................................................... 45
    2.5.2. Continuity Equation Linear Solution Strategy ....................................... 47
    2.5.3. Coarsening Strategies ............................................................................. 47
  2.6. Interface Sliding Technique for Rotor-Stator Computation ......................... 49
    2.6.1. Interface Strategy ................................................................................... 49
    2.6.2. Transfer of Information across Interface .............................................. 54
    2.6.3. Velocity Components ............................................................................. 56
    2.6.4. Gradient .................................................................................................. 57
    2.6.5. Second Order Scheme for the Momentum Equations ............................ 58
  2.7. Frontend and Backend for Code B .................................................................. 58
  2.8. Parallelization .................................................................................................. 60
CHAPTER 3. EFFECTS OF BLADE SPACING AND WAKE/BLADE COUNT RATIO ON THE UNSTEADY FLOW IN A COMPRESSOR STATOR .................................................69

3.1. Simulation Test Case ...........................................................................................70
3.2. Effect of Axial Spacing between Blade Rows.....................................................72
   3.2.1. Unsteady Blade Pressure and Velocity Field ............................................73
   3.2.2. Steady and Unsteady Blade Boundary Layer............................................76
   3.2.3. Rotor Wake Decay and Loss Coefficient ................................................80
3.3. Effect of Wake/Blade Count Ratio ......................................................................80
   3.3.1. Steady and Unsteady Boundary Layer and Losses.................................82

CHAPTER 4. THREE DIMENSIONAL FLOW IN A HIGH REYNOLDS NUMBER PUMP INLET GUIDE VANE. .......................................................................................104

4.1. Case Description................................................................................................104
4.2. Flow Field inside the IGV Passage....................................................................104
   4.2.1. Static Pressure Distribution on the Blade Surface...................................104
   4.2.2. Momentum Thickness and Loss Coefficient ...........................................106
4.3. Flow at the Exit of the IGV ...............................................................................108
   4.3.1. Radial Distribution of the Passage-Averaged Velocities .........................108
   4.3.2. Wake Characteristics and Effects of Turbulence Length Scale ..............109
   4.3.3. Secondary Flow at the Exit.................................................................111
   4.3.4. The Decay Characteristics of the Wake and the Secondary Flow..........112

CHAPTER 5. FULLY COUPLED ROTOR-STATOR INTERACTION IN AN AXIAL TURBINE STAGE. .........................................................................................................134

5.1. A Test Case: Vortex Convection .......................................................................134
5.2. Case Description.................................................................................................136
5.3. Grid Generation and Overall Flow Field...........................................................138
   5.3.1. Unstructured Grid for AFTRF Stage.......................................................139
   5.3.2. Overall Flow Field for the AFTRF Stage................................................142
5.4. Nozzle Flow Field.............................................................................................146
   5.4.1. Flow inside the Nozzle Blade Passage ....................................................146
   5.4.2. Nozzle Wakes..........................................................................................147
5.5. Rotor Flow Field................................................................................................150
   5.5.1. Static Pressure Distribution on the Rotor Blade Surface .........................150
   5.5.2. Flow inside the Rotor Blade Passage ......................................................153
   5.5.3. Nozzle Wakes inside Rotor Blade Passage .............................................153
   5.5.4. Rotor Wakes ............................................................................................160

CHAPTER 6. SINGLE-PHASE AND MULTIPHASE ROTOR-STATOR ANALYSIS OF FLOW IN AN AXIAL PUMP.........................................................................................214

6.2. Case Description and Grid Generation...............................................................215
6.4. Multi-Phase Rotor-Stator Interaction ................................................................. 217

CHAPTER 7. CONCLUDING REMARKS AND RECOMMENDATIONS FOR FUTURE RESEARCH .......................................................... 228
  7.1. Concluding Remarks ...................................................................................... 228
  7.2. Recommendations for Future Research ...................................................... 230

REFERENCES ........................................................................................................ 232
LIST OF FIGURES

Figure 2.1. Control volume with grid point notations for code A .................................61
Figure 2.2. Arbitrary polyhedral element and hierarchal data structure for code B .........62
Figure 2.3. Geometry nomenclature for cell face evaluations .......................................63
Figure 2.4. Coarsening of grid cells, (a) top: isotropic; (b) bottom: directional ..........64
Figure 2.5. Class diagram for AMG solver .................................................................65
Figure 2.6. Elements of the frontend for code B ..........................................................66
Figure 2.7. Sketches for interface sliding technique .....................................................67
Figure 2.8. Sketches for interface sliding technique at time step 1 ...............................68
Figure 3.1. Inlet velocity profiles for cases 1 to 4 (linear interpolation from experimental data by Stauter, et al., 1991) at 36% axial chord upstream of the stator) ..........84
Figure 3.2. Rotor wake profile at 13% axial chord upstream of the stator. symbol: experimental data by Stauter, et al. (1991); line: prediction ....................................................85
Figure 3.3. The second stage rotor and stator blade setting ...........................................86
Figure 3.4. The first Fourier component of the suction surface unsteady pressure distribution, C_p1, (a) magnitude; (b) phase angle .................................................................87
Figure 3.5. Predicted unsteady fluctuations of the velocity vector, U(t)-U_0. (a) case 1: s/C_x = 0.80 ................................................................................................................88
Figure 3.5 (Cont.). Predicted unsteady fluctuations of the velocity vector, U(t)-U_0. (b) case 3: s/C_x = 0.30 ........................................................................................................89
Figure 3.6. The predicted unsteady fluctuations of the vorticity, \Omega(t) - \Omega_0. (a) case 1: s/C_x = 0.80 ........................................................................................................90
Figure 3.6 (Cont.). The predicted unsteady fluctuations of the vorticity, \Omega(t) - \Omega_0. (b) case 3: s/C_x = 0.30 ........................................................................................................91
Figure 3.7. The second Fourier component of the suction surface unsteady pressure distribution, C_p2, (a) magnitude; (b) phase angle .................................................................92
Figure 3.8. The first Fourier harmonic of the unsteady total blade force as a function of the spacing, s/C_x, (a) axial component; (b) tangential component. symbol: experimental data by Gallus, et al. (1982); line: prediction .........................................................................93
Figure 3.9. The predicted momentum thickness on the suction surface, \theta/C_x, (a) time mean, (b) first harmonic, (c) second harmonic .................................................................................94
Figure 3.10. The predicted skin friction coefficient on the suction surface, C_f, (a) time mean, (b) first harmonic, (c) second harmonic .................................................................................95
Figure 3.11. Decay characteristic of the upstream rotor wake, (a) axial velocity defect; (b) maximum relative tangential velocity variation across the wake ................................................................................96
Figure 3.12. Distribution of the loss coefficient, (a) upstream of the stator leading edge; (b) inside the stator blade passage ..................................................................................97
Figure 3.13. Loss coefficient at the exit of the stator as a function of spacing, s/C_x ......98
Figure 3.14. First Fourier component of the suction surface unsteady pressure distribution, C_p1, (a) magnitude; (b) phase angle .................................................................................99
Figure 3.15. Unsteady velocity vector, U(t)-U_0; (a) case 6: r = 2.0 at t/T = 0.15 ..........100
Figure 3.15 (Cont.). Unsteady velocity vector U(t)-U_0; (b) case 7: r = 3.0 at t/T = 0.15 ....101
Figure 3.16. Momentum thickness on the suction surface, \theta/C_x, (a) time mean, (b) first
harmonic, (c) second harmonic......................................................................................................102
Figure 3.17. Distribution of the loss coefficient, $C_f$: (a) upstream of the stator leading edge, (b) inside the stator blade passage. ........................................................................................103
Figure 4.1. Static pressure distribution on the blade surface of the IGV (90% span is near the tip). symbol: experimental data by Zierke, et al. (1993); line: prediction. .........................114
Figure 4.2. Momentum thickness distribution on the suction surface of the IGV. ..........115
Figure 4.3. Distribution of loss coefficient inside and downstream of the blade passage of the IGV. .......................................................................................................................116
Figure 4.4. Passage-averaged velocity distribution at 49.7%Cx downstream of the IGV trailing edge. symbols: experimental data by Zierke, et al. (1993); lines: predictions. ....117
Figure 4.5. Passage-averaged yaw angle at 49.7% axial chord downstream of the IGV trailing edge. symbols: experimental data by Zierke, et al. (1993); lines: predictions. ....118
Figure 4.6. Wake profiles at 49.7% axial chord downstream of the IGV trailing edge. (a) 9.5% span. symbols: experimental data by Zierke, et al. (1993); lines: predictions. ......119
Figure 4.6 (Cont.). Wake profiles at 49.7%Cx downstream of the IGV trailing edge. (b) 52.4% span. symbols: experimental data by Zierke, et al. (1993); lines: predictions. ....120
Figure 4.6 (Cont.). Wake profiles at 49.7%Cx downstream of the IGV trailing edge. (c) 9.5% span. symbols: experimental data by Zierke, et al. (1993); lines: predictions. ......121
Figure 4.7. Decay characteristics of the wake at mid-span due to simulations. ..........122
Figure 4.8. Loss coefficient distribution at the mid-span due to simulations. ..........123
Figure 4.9. Fourier content of the wake at 49.7%Cx downstream of the IGV trailing edge. (a) 9.5% span. symbols: experimental data by Zierke, et al. (1993); lines: predictions. .124
Figure 4.9 (Cont.). Fourier content of the wake at 49.7% axial chord downstream of the IGV trailing edge. (b) 52.4% span. symbols: experimental data by Zierke, et al. (1993); lines: predictions. ......125
Figure 4.9 (Cont.). Fourier content of the wake at 49.7% axial chord downstream of the IGV trailing edge. (c) 81.0% span. symbols: experimental data by Zierke, et al. (1993); lines: predictions. ..................................................126
Figure 4.10. Secondary velocity vectors ($v_s$) at 49.7% chord axially downstream of the IGV trailing edge. (a) Prediction. ..........................................................127
Figure 4.10 (Cont.). Secondary velocity vectors ($v_s$) at 49.7% chord axially downstream of the IGV trailing edge. (b) experimental data by Zierke, et al. (1993) .........................128
Figure 4.11. Flow deviation angle ($\Delta \alpha_2$ in degrees) distribution at 49.7% axial chord downstream of the IGV trailing edge. (a). Prediction.................................129
Figure 4.11 (Cont.). Flow deviation angle ($\Delta \alpha_2$ in degrees) distribution at 49.7%Cx downstream of the IGV trailing edge. (b) experimental data by Zierke, et al. (1993) ....130
Figure 4.12. Contour of the axial velocity defect ($v_{x,d}$) in the x-r plane. ..................131
Figure 4.13. X-r contour of the maximum variation in the tangential velocity ($v_{q,d}$). 132
Figure 4.14. Comparison of the decays of the wake and the secondary flow..........133
Figure 5.1. Vortex vector (maximum length in figure = 0.289) and pressure contour (blue = -0.10 and red = 0.01) for the Stokes vortex case 1. ..................................................162
Figure 5.2. Profiles of the vortex core pressure and the maximum rotating velocity for the Stokes vortex with different grid size. ..................................................163
Figure 5.3. Mid-section view of the unstructured grid with 4 nozzle and 5 rotor blades for the AFTRF configuration.................................................................................................164
Figure 5.4. Mid-section view of the unstructured grid in the nozzle wake and interface regions for the AFTRF configuration. ..................................................................................................................165
Figure 5.5. Pseudo-time convergence history of the axial velocity and pressure for five successive physical time steps for the AFTRF rotor-stator computation. .................................166
Figure 5.6. Instantaneous distribution of the static pressure coefficient (Cp) at t/T=0.0 for the AFTRF rotor-stator computation..........................................................167
Figure 5.7. Instantaneous distribution of the static pressure coefficient (Cp) near the interface region at t/T=0.0 for the AFTRF rotor-stator computation ........................................168
Figure 5.8. Instantaneous distribution of the axial velocity (v/Um) near the interface region at t/T=0.0 for the AFTRF rotor-stator computation................................................169
Figure 5.9. Instantaneous distribution of the relative tangential velocity (wq/Um) near the interface region at t/T=0.0 for the AFTRF rotor-stator computation ........................................170
Figure 5.10. Time-averaged relative flow angle (β in degrees) for the AFTRF rotor-stator unsteady computation. ..................................................................................................171
Figure 5.11. Time-averaged absolute total pressure coefficients (Cp,tot) for the AFTRF rotor-stator computation ..................................................................................................172
Figure 5.12. Time-averaged absolute total pressure coefficient (Cp,tot) at the interface and rotor exit for the AFTRF rotor-stator computation ........................................................................173
Figure 5.13. Time-averaged static pressure coefficients on the nozzle blade surfaces. symbol: experimental data by Zaccaria (1994); line: prediction ................................174
Figure 5.14. Unsteady fluctuations of the static pressure coefficients (Cp(t)-Cp) on the nozzle blade surface for the AFTRF rotor-stator computation. ......................................................175
Figure 5.15. The first three harmonics of the static pressure coefficients on the nozzle blade surface for the AFTRF rotor-stator computation. .................................................................176
Figure 5.16. Time-averaged absolute total velocity distribution near the nozzle mid chord at x/Cx=0.56 for the AFTRF rotor-stator computation. symbols: experimental data by Zaccaria (1994) ..................................................................................................................................177
Figure 5.17. The time-averaged profiles of the nozzle wake at five axial stations downstream of the nozzle trailing edge for the AFTRF rotor-stator computation. symbols: experimental data by Zaccaria (1994); line: prediction ..........................................................178
Figure 5.18. The time-averaged profiles of the nozzle wake with grids shown for the AFTRF rotor-stator computation. ..................................................................................................179
Figure 5.19. Spatial harmonic content of the nozzle wake at four axial locations downstream of the nozzle for the AFTRF rotor-stator computation. symbols: experimental data by Zaccaria (1994); lines: prediction ..........................................................180
Figure 5.20. Decay of the spatial harmonic content of the nozzle wake for the AFTRF rotor-stator computation. Top: the first harmonic; Bottom: the second and third harmonics; symbols: experimental data by Zaccaria (1994); lines: prediction ..........................................................181
Figure 5.21. The time-averaged profiles of the nozzle wake absolute flow angle at three axial stations downstream of the nozzle trailing edge for the AFTRF rotor-stator computation. symbols: experimental data by Zaccaria (1994); lines: prediction ..........182
Figure 5.22. The time-averaged profiles of the nozzle wake static pressure at $x/C_x = 1.025$ for the AFTRF rotor-stator computation. symbols: experimental data by Zaccaria (1994); lines: prediction................................................................................................................183

Figure 5.23. Predicted time-averaged static pressure distribution around the nozzle exit for the AFTRF rotor-stator computation..........................................................184

Figure 5.24. Time-averaged and unsteady envelope of the static pressure distribution on the rotor blade surface for the AFTRF rotor-stator computation. symbols: experimental data by Lakshminarayana, et al. (2000); lines: prediction........................................185

Figure 5.25. Predicted unsteady fluctuations of the static pressure distribution ($C_p(t)-C_p$) on the rotor blade surface for the AFTRF rotor-stator computation..........................................................186

Figure 5.26. Distribution of the first three harmonics of the static pressure on the rotor blade surface for the AFTRF rotor-stator computation. symbols: experimental data by Lakshminarayana, et al. (2000); lines: prediction..........................................................187

Figure 5.27. Blade-to-blade distributions of the total relative velocity ($W/U_m$) at three axial locations inside the rotor blade passage for the AFTRF rotor-stator computation. symbols: experimental data by Zaccaria (1994)....................................................................................188

Figure 5.28. Mass-averaged total absolute velocity ($V/U_m$) through the rotor blade passage for the AFTRF rotor-stator computation. symbols: experimental data by Zaccaria (1994); line: prediction .................................................................................................................189

Figure 5.29. Mass-averaged relative flow angle ($\beta$) through the rotor blade passage for the AFTRF rotor-stator unsteady computation. symbol: experimental data by Zaccaria (1994); line: prediction .................................................................................................................190

Figure 5.30. Snapshot of the turbulence intensity ($T_u$) at nozzle position 1. top: experimental data by Zaccaria (1994); bottom: prediction (contour interval is 2). ........191

Figure 5.30 (Cont.). Snapshot of the turbulence intensity ($T_u$) at nozzle position 2. top: experimental data by Zaccaria (1994); bottom: prediction (contour interval is 2). ........192

Figure 5.30 (Cont.). Snapshot of the turbulence intensity ($T_u$) at nozzle position 3. top: experimental data by Zaccaria (1994); bottom: prediction (contour interval is 2). ........193

Figure 5.30 (Cont.). Snapshot of the turbulence intensity ($T_u$) at nozzle position 4. top: experimental data by Zaccaria (1994); bottom: prediction (contour interval is 2). ........194

Figure 5.30 (Cont.). Snapshot of the turbulence intensity ($T_u$) at nozzle position 5. top: experimental data by Zaccaria (1994); bottom: prediction (contour interval is 2). ........195

Figure 5.30 (Cont.). Snapshot of the turbulence intensity ($T_u$) at nozzle position 6. top: experimental data by Zaccaria (1994); bottom: prediction (contour interval is 2). ........196

Figure 5.31. Snapshot of the total relative velocity ($W/U_m$) at nozzle position 1. Top: experimental data by Zaccaria (1994); Bottom: unsteady prediction, where contour value interval is 0.11..................................................................................................................197

Figure 5.31 (Cont.). Snapshot of the total relative velocity ($W/U_m$) at nozzle position 2. Top: experimental data by Zaccaria (1994); Bottom: prediction, where contour value interval is 0.11..................................................................................................................198

Figure 5.31 (Cont.). Snapshot of the total relative velocity ($W/U_m$) at nozzle position 3. Top: experimental data by Zaccaria (1994); Bottom: prediction, where contour value interval is 0.11...
Figure 5.31 (Cont.). Snapshot of the total relative velocity ($W/U_m$) at nozzle position 4.
Top: experimental data by Zaccaria (1994); Bottom: prediction, where contour value interval is 0.11.

Figure 5.31 (Cont.). Snapshot of the total relative velocity ($W/U_m$) at nozzle position 5.
Top: experimental data by Zaccaria (1994); Bottom: prediction, where contour value interval is 0.11.

Figure 5.31 (Cont.). Snapshot of the total relative velocity ($W/U_m$) at nozzle position 6.
Top: experimental data by Zaccaria (1994); Bottom: prediction, where contour value interval is 0.11.

Figure 5.32. Snapshot of the predicted total relative velocity fluctuations ($W'/U_m$) at nozzle position 1.

Figure 5.33. Snapshot of the predicted total absolute velocity fluctuations ($V'/U_m$) at nozzle position 1.

Figure 5.34. Snapshot of the relative flow angle at nozzle position 1. Top: experimental data by Zaccaria (1994); Bottom: prediction, where contour value interval is 7.5 deg.

Figure 5.34 (Cont.). Snapshot of the relative flow angle at nozzle position 2. Top: experimental data by Zaccaria (1994); Bottom: prediction, where contour value interval is 7.5 deg.

Figure 5.34 (Cont.). Snapshot of the relative flow angle at nozzle position 3. Top: experimental data by Zaccaria (1994); Bottom: prediction, where contour value interval is 7.5 deg.

Figure 5.34 (Cont.). Snapshot of the relative flow angle at nozzle position 4. Top: experimental data by Zaccaria (1994); Bottom: prediction, where contour value interval is 7.5 deg.

Figure 5.34 (Cont.). Snapshot of the relative flow angle at nozzle position 5. Top: experimental data by Zaccaria (1994); Bottom: prediction, where contour value interval is 7.5 deg.

Figure 5.34 (Cont.). Snapshot of the relative flow angle at nozzle position 6. Top: experimental data by Zaccaria (1994); Bottom: prediction, where contour value interval is 7.5 deg.

Figure 5.35. Time-averaged blade-to-blade distributions of the total relative velocity ($W/U_m$) for the rotor wakes at five axial locations for the AFTRF rotor-stator computation, symbols: experimental data by Zaccaria (1994); lines: prediction.

Figure 5.36. Time-averaged blade-to-blade distributions of the relative flow angle for the rotor wakes at five axial locations for the AFTRF rotor-stator computation. symbols: experimental data by Zaccaria (1994); lines: prediction.

Figure 5.37. Distributions of fluctuations of relative total velocity at five axial locations for the AFTRF rotor-stator computation.

Figure 6.1. Views of hybrid grid used for HIREP rotor-stator computation.

Figure 6.2. Pseudo-time convergence history for five successive physical time steps for HIREP rotor-stator computation.

Figure 6.3. Time snapshots of predicted axial velocity ($u/U_{tip}$) at $t/T=0.00$, 0.25, 0.50, 0.75 (top to bottom) for HIREP stage.
Figure 6.4. Predicted and experimental time averaged IGV midspan blade pressure distributions for HIREP configuration. symbol: experimental data by Zierke, et al. (1993); lines: prediction.................................................................223
Figure 6.5. Predicted and experimental time averaged rotor midspan blade pressure distributions for HIREP configuration. symbol: experimental data by Zierke, et al. (1993); lines: prediction.................................................................224
Figure 6.6. Predicted first five harmonics of the unsteady pressure on the rotor blade surface for the HIREP rotor-stator computation. ........................................................................225
Figure 6.7. Snapshots of predicted field 2(20µm) volume fraction at t/T=0.0, 0.25, 0.5, 0.75 (top to bottom) for HIREP stage with simulated hull bubble boundary layer inflow. 6-field, 2:1 stage computation. Range: red= .002, green= .001. ...................................................226
Figure 6.8. Snapshots of predicted $\alpha^4-\alpha^2$ (100, 20µm repectively) volume fraction at t/T=0.0, 0.25, 0.5, 0.75 (top to bottom) for HIREP stage with simulated hull bubble boundary layer inflow. 6-field, 2:1 stage computation. Range: red= .0003, blue= -.0003. ............227
LIST OF TABLES

Table 3.1. Test cases for simulations of blade row spacing effects.................................71
Table 3.2. Test cases for simulations of wake/blade count ratio ....................................72
Table 5.1. Core pressure and maximum rotating velocity of the Stokes vortex................136
### NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Control face; Area; Coefficient</td>
</tr>
<tr>
<td>C</td>
<td>Chord</td>
</tr>
<tr>
<td>$C_f$</td>
<td>Skin friction coefficient</td>
</tr>
<tr>
<td>$C_p$</td>
<td>Static pressure coefficient</td>
</tr>
<tr>
<td>$C_{p,\text{tot}}$</td>
<td>Total pressure coefficient</td>
</tr>
<tr>
<td>$C_r$</td>
<td>Axial chord of rotor</td>
</tr>
<tr>
<td>$C_x$</td>
<td>Axial chord, axial chord of nozzle</td>
</tr>
<tr>
<td>e</td>
<td>Unit vector</td>
</tr>
<tr>
<td>F</td>
<td>Body force; aerodynamic force</td>
</tr>
<tr>
<td>f</td>
<td>Grid face</td>
</tr>
<tr>
<td>$G_1, G_2, G_3$</td>
<td>Contravariant velocity components</td>
</tr>
<tr>
<td>J</td>
<td>Jacobian of coordinates transformation</td>
</tr>
<tr>
<td>$K_p$</td>
<td>Static pressure coefficient</td>
</tr>
<tr>
<td>k</td>
<td>Turbulence kinetic energy</td>
</tr>
<tr>
<td>$l_t$</td>
<td>Turbulence length scale</td>
</tr>
<tr>
<td>$\dot{m}$</td>
<td>Mass flow rate</td>
</tr>
<tr>
<td>$N_n$</td>
<td>Number of nozzle blades in a row</td>
</tr>
<tr>
<td>$N_r$</td>
<td>Number of rotor blades in a row</td>
</tr>
<tr>
<td>nb</td>
<td>Neighboring points</td>
</tr>
</tbody>
</table>
P  Total pressure
p  Static pressure
p'  Pressure correction
R  Rotation operator
r  radius; ratio of blade numbers
Re  Reynolds number
S  Source term
s  Axial gap, nondimensionalized pitchwise distance
T  Unsteady period
T_n  Unsteady period for stator passage
T_r  Unsteady period for rotor passage
t  Time
Δt  Physical time step
Tu  turbulence intensity
U  Total velocity; total absolute velocity
U_m  Blade rotation speed
U_{tip}  Blade rotation speed at tip
u, v, w  Velocity components
V  Total absolute velocity
V_c  Total velocity at wake center
W  Total relative velocity
x  Axial coordinate, axial coordinate from nozzle leading edge

\(x_r\)  Axial distance from rotor leading edge

y, z  Cartesian coordinates

\(y^+ = \rho y \sqrt{\tau_w/\rho/\mu}\)

Greek

\(\alpha\)  Absolute flow angle

\(\alpha, \beta, \gamma\)  \(\nabla \xi \cdot \nabla \xi, \nabla \eta \cdot \nabla \eta, \nabla \zeta \cdot \nabla \zeta\)

\(\beta\)  Relative flow angle

\(\Gamma\)  Isotropic diffusion coefficient

\(\phi\)  General scalar; phase angle

\(\varepsilon\)  Turbulence dissipation rate

\(\varepsilon_2, \varepsilon_4\)  Second- and fourth-order artificial dissipation coefficients

\(\varepsilon_{pw}\)  Fourth order artificial dissipation coefficient for pressure

\(\theta\)  Circumferential coordinate; momentum thickness

\(\theta_1, \theta_2, \theta_3\)  \(\nabla \xi \cdot \nabla \eta, \nabla \xi \cdot \nabla \zeta, \nabla \eta \cdot \nabla \zeta\)

\(\mu\)  Molecular viscosity

\(\mu_t\)  Turbulent eddy viscosity

\(\zeta\)  Total pressure loss coefficient

\(\rho\)  Density

\(\Omega\)  Vorticity
\[ \tau \] Stress tensor

\( \tau_w \) Wall shear stress

\( \omega \) Angular velocity; relaxation parameter

\( \omega_r \) Reduced frequency

\( \xi, \eta, \zeta \) Generalized coordinates

Subscripts

\( \infty \) Upstream value

\( 0 \) Time-averaged value

\( 1 \) Inlet

\( 1,2,3 \) Order of harmonic

\( d \) Maximum defect

\( E, W, N, S, T, B \) Grid points to the east, west, north, south, top, bottom

\( e, w, n, s, t, b \) Control face to the east, west, north, south, top, bottom

\( f \) Face

\( e \) Freestream value

exit Exit value

\( i, j, k \) Components of a vector or scalar; grid index

\( \text{in} \) Inlet

\( \text{le, te} \) Leading edge, trailing edge

\( n \) Order of harmonic

\( \text{ref} \) Reference point
\( \theta \) Circumferential component

\( r \) Radial component

**Superscripts**

' Turbulent fluctuation; instantaneous fluctuation

- Time-average; Mass-average

n Time step
ACKNOWLEDGEMENTS

The author owes a special debt of gratitude to his thesis advisor, Dr. R.F. Kunz, for his outstanding guidance and direct assistance in helping with this thesis. His hands on approach, hard-working attitude, and frequent daily interactions in working on the code extension and related cases were invaluable in this thesis research. The author would also like to thank the other members of the thesis committee, co-chair Dr. P.J. Morris, Dr. L.N. Long, Dr. V. Yang, and Dr. B. Lakshminarayana, for their guidance, invaluable advice, suggestions, comments, and encouragement throughout the course of this study. A special thanks goes to department head, Dr. D.K. McLaughlin, for his help and encouragement during the course of this study.

This thesis would not have been finished without the love, support, patience, and understanding of the author’s family.

This work was in part supported by the Office of Naval Research under contract No. N00014-90-J1182 with J.Fein as the contract monitor. The author also would like to acknowledge Center for Academic Computing at Penn State, NASA’s NAS Facility at NASA Ames Research Center, and Pittsburgh Supercomputing Center for providing computing resources.
CHAPTER 1 INTRODUCTION

1.1 Background and Justification

Turbomachines are widely used for power and propulsion in automotive (cars and trucks), marine (ships and submarines), aeronautical (gas turbine), and space vehicles (liquid rockets); biomedical (artificial and assist heart pumps); power (steam, nuclear, and cogeneration), and pumping industry. Energy consumption due to turbomachinery represents a substantial part of the total energy consumption worldwide. The ability to design turbomachinery with good efficiency and reliability over the whole operating range for a particular application is therefore of great economic significance. A fundamental component of such ability is the understanding of physical mechanisms involved in the fluid flows through turbomachines.

The flow and thermal processes occurring in turbomachinery blade rows are very complex. The flow field is three-dimensional and unsteady, with laminar, transitional and turbulent flow on the blade profiles. Heat transfer rates are controlled by the nature of these viscous layers. The flow and thermal fields in the hub wall regions are controlled by the presence of a horseshoe vortex, secondary flow and corner flow separation. The interaction of these flow patterns with the blade boundary layers produces many complex flow features. The presence of leakage flow, secondary flow, the horseshoe vortex and annulus wall and blade boundary layers in the endwall region makes the endwall flow highly three-dimensional and turbulent with appreciable variation (from design) of heat transfer and flow field.
It is well known that the flow field in a turbomachinery stage is unsteady. In fact, it is precisely the unsteady process inside a turbomachinery blade row that makes the extraction or addition of energy from the working fluid possible. The unsteadiness, due to relative motion between a stator blade and a rotor blade, has a major influence on the aerodynamic, mechanical and thermal performance as well as the cooling requirements for turbines. The mechanical reliability, life and cycle performance are all substantially altered due to these interactions.

The unsteadiness in the turbomachinery flow field is caused by several phenomena, which are listed below.

(1) The potential (pressure) interaction between a rotor and a stator. This will be substantial for small rotor-stator spacing and large blade loading.

(2) The viscous interaction. This is due to the velocity gradients in blade wakes impinging on a subsequent blade row. This is perhaps one of the major contributors to flow unsteadiness.

(3) The random unsteadiness in the free stream, present even in an otherwise steady flow. It results in random unsteady pressure on the blade. The free stream turbulence intensity, length scale and structure as well as their interaction with a nozzle wake and their subsequent transport through a rotor is known to have a major influence on the rotor blade boundary layer and passage flow.

(4) Unsteadiness due to the presence of skewed boundary layers on end walls and tip leakage in a rotor. It has substantial unsteady effects in the succeeding blade rows.
(5) The unsteady flow field induced by blade vibration and the fluid-structure interaction. It could result in blade failure in many cases and is commonly known as flutter.

(6) An additional source of unsteadiness in a turbine rotor due to the secondary flow in nozzle passages resulting from the radial and circumferential temperature gradient present at the exit of a combustion chamber.

(7) A number of indirect effects, such as change in transition, stagnation point, influence of the downstream pressure gradient on the wake decay, and the trailing vortex strength.

(8) Unsteadiness due to shock waves. In high-speed rotors, the trailing edge shock system will generate additional unsteady interactions. If the inlet Mach number is supersonic in the downstream blade row, the unsteady shock system generated due to an upstream wake will introduce substantial unsteadiness and vibration.

There are a wide variety of parameters contributing to the complexity of unsteady aerodynamics in turbomachines: Reynolds number, Mach number, rotation number, boundary layer parameters at the trailing edge, free stream turbulence intensity and length scales, blade and vane turning angle, blade profile including trailing edge thickness, chordwise pressure distribution, Richardson number representing curvature effect, blade aspect ratio, blade spacing, inlet flow boundary layer profile and thickness, tip clearance height, parameters characterizing the wake width, profile and decay, rotor-stator spacing, etc.
With the trend toward compact size and high loading in the design of turbomachinery, the flow unsteadiness becomes more and more severe. The unsteadiness causes blade vibration and noise generation. It also affects loss generation, efficiency, and heat transfer due to wake-induced transition. The blade vibration may cause unacceptable levels of stress and shorten the blade fatigue life. It could be disastrous if the vibrating frequency matches the blade natural frequency. The undesirable aspects of the unsteady flows should be controlled to ensure reliable, safe and quiet operation of the turbomachines.

The high performance and durability of turbomachinery can be realized by an improved understanding of the physics associated with the interaction phenomena. Considerable attention has been paid in recent years to the development of computer codes to predict unsteady aerodynamics and heat transfer, but these efforts are hampered by a lack of understanding of the basic physics associated with these interactions and the lack of adequate physical modeling (transition/turbulence models) and validation of the codes.

The leakage flow creates a loss in efficiency primarily because the blade unloading in the tip leads to a decrease in the relative pressure difference. This could be two to four percent of the overall efficiency of the turbomachine. The leakage flow also creates a flow discontinuity because the leakage jet and the main flow are at different angles. This discontinuity will eventually roll up into the core of a vortex. Because of the relative motion between the rotor and the stator, the leakage vortex is seen as unsteadiness by the succeeding stator. This causes unsteady pressures, boundary layers, and transition, all of which can induce vibrations and lead to noise generation.
Experimental study of the flow is limited by measurement techniques. Furthermore, detailed experimental investigations of a particular geometry for a turbomachine require sophisticated manufacturing technologies and a large financial investment and time. So, experimental study is greatly limited to a narrow operation range. Mathematical models have long been recognized as a more economical approach in understanding the flow mechanisms in turbomachinery. The Navier-Stokes equations, in describing the dynamics of fluid flow, provide the basis for mathematical models of fluid flow. Due to the complexity of the flow, any analytical or simplified models of the Navier-Stokes equations can only be used to study very idealized flow structures.

Currently, most of the aerodynamic prediction methods used in the aeroelastic and aeroacoustic design are based on classical linearized inviscid theory (Whitehead, 1960). This theory is only valid for lightly loaded thin-airfoil cascades. In the past twenty years, progress has been made toward using the three-dimensional Euler and Navier-Stokes methods (Verdon, 1993). Most of the work uses a time-marching method for compressible flows.

Numerical models of the full governing equations are, however, expected to capture the details of the complex flow structures in turbomachinery. Computational Fluid Dynamics (CFD) has become an increasingly applied and accepted tool in the analysis and design of turbomachinery. With the advances in algorithms and computer architecture, accurate three-dimensional flow predictions can be performed in a reasonable amount of time. While a sizable simulation can incur a substantial cost in computational resources, a clear understanding of the aerodynamic processes aids the design of turbomachinery con-
siderably. This thorough comprehension of the entire turbomachine flow field can lead to
greatly enhanced performance and efficiency. Thus, the computational costs incurred are
completely justified.

Future design toward lighter, safer, quieter, and more durable turbomachines needs
a better understanding of the various unsteady interactions and the mechanism of the
effects of these interactions on the turbomachinery performance. Numerical computation
of these unsteady interactions is a very powerful tool in achieving this understanding. In
the past twenty years, a lot of computational work has been done on this subject. However,
a lot of work still needs to be carried out in terms of validating the unsteady flow code,
in incorporating more advanced turbulence models, and having the ability to calculate realis-
tic multi-stage turbomachines.

1.2 Objectives and Scope

The understanding of three-dimensional and unsteady flows requires an aerody-
namic prediction method, which is not only the solution to resolve the unsteady three-
dimensional flows, but also serves as a basis for the aeroelastic and aeroacoustic analyses.
The major objective of this research is to understand the various sources of unsteadiness in
two/three-dimensional flows through turbomachinery and provide new insights into the
effects of these sources on the turbomachinery performance. This goal will be achieved
through the modification and applications of two adapted CFD codes that are capable of
predicting three-dimensional unsteady viscous flows, with one of the codes being able to
compute multiphase flows.
The main emphasis will be in capturing the physics of the flow, including the wake, unsteady and steady pressures, the three-dimensional boundary layers on annulus and hub walls, and the transport of wakes and secondary flows through the passage. The investigation will include new techniques developed for efficient and accurate transfer of information between blade rows. The code can then be used to understand and resolve the sources and magnitude of unsteady flow, pressure, and shear stress, and correlate it with the inlet distortion due to nozzle wakes and secondary flow. A thorough understanding of the flow physics will lead to methods of controlling these undesirable features through flow modification.

One of the codes adapted in this research is a three-dimensional pressure-based Navier-Stokes solver developed by Basson (1992) and later extended by Ho (1995). It will be used to predict the steady and unsteady turbulent flows. The code uses a SIMPLE (Semi-Implicit Methods for Pressure Linked Equations, Patankar, 1980) type method. Implicit schemes are used for each of the momentum equations. A pressure equation is derived from the continuity equation and is also solved implicitly. All these equations are solved separately, thus avoiding the large computational time involved in the inversion of large matrices. A predictor-corrector type algorithm is applied to the governing equations to ensure the time-accuracy. The code has been used successfully for solving incompressible flows and proved to be efficient and accurate.

The second code adapted in this research is developed by Kunz, et al. (2001). It is also a pressure-based Navier-Stokes solver, but it has several new and modern elements in it. Unstructured grids are used in the code to account for the complex geometry common
in turbomachines. They have the advantage of the relatively easy implementation of a fine mesh in the region of the wake, both from its originating blade and subsequent blade rows. This will ensure the transport of the wake is well resolved spatially. The first code can only calculate a single blade row. The wake passing strategy used ensures the wake/blade interaction is accounted for, but the potential interaction between blade rows is omitted. Rotors and stators are solved together in the second code; hence it more accurately models the unsteady interactions. Parallel processing capability and full two-fluid modeling for multiphase flow analysis are also implemented in the code.

Another objective of this research is to gain new insights into the physics of three-dimensional unsteady flows. Topics such as the effects of blade rows spacing, number of wakes, tip clearance vortex, secondary flows and turbulence effects will be explored during this research.

In turbomachinery flows, wakes decay when they travel downstream in the axial gap between the rotor and the stator because the flows are viscous. The amplitude of the wake depth decreases and the wake width increases when it reaches the downstream blade row. The rate of this decay is high near the blade trailing edge where the wake originates and decreases along the flow direction. Certain correlations between the wake decay rate and the axial coordinates have been developed (e.g., Raj and Lakshminarayana, 1973). However, the correlations may not hold for a close axial gap because of the strong unsteady interaction between the wake and the downstream blade rows. This interaction is usually neglected in the single-blade row approach, but can be resolved in a multi-blade row approach. On one hand, the closer the axial gap, the stronger the wake would be when
it reaches the downstream blade. But on the other hand, the unsteady interaction usually
tends to augment the decay of the wake and hence less unsteady pressure would be generated
on the subsequent blade surfaces. It is interesting to know which effect is more domi-
nant. Also, the axial gap can have effects on the loss generation mechanism in the
subsequent blade passages.

The wake/blade count ratio effects can be divided into two categories: the effect of
reduced frequency and the effect of increased interaction between the wakes in one down-
stream blade passage. Higher wake/blade count ratio results in higher reduced frequency
and hence smaller unsteadiness in the subsequent blade rows. Higher interaction between
the wakes leads to the faster decay of the wake and smaller unsteadiness.

Besides the unsteadiness due to the wakes, secondary flow and tip clearance vorti-
ces can be also major sources of generating unsteadiness in the downstream blade rows. It
is found that the decay of the secondary flows and tip vortex is slower in the endwall
region than the wakes (Yu, et al. 1996). The magnitude of the flow non-uniformity can be
higher due to secondary flows and tip vortex than the wakes. Clarification of these sources
of unsteadiness and evaluate their individual contributions to the flow unsteadiness can be
very valuable in the design process.

In summary, the objectives of this thesis are as follows: (1) To extend two CFD
codes to calculate the three-dimensional, steady and unsteady viscous flows through tur-
bomachinery blade rows, including multistage blade rows. (2) To predict unsteady flows
due to rotor-stator interaction with detailed analysis of the effects of the unsteadiness on
the blade boundary layer development and wake decay. (3) To assess the effects of the
rotor-stator spacing and wake/blade count ratio on the unsteady flow. (4) To predict a two-phase rotor-stator interaction to gain insight into the physical mechanism of unsteady bubbly flow.

1.3 Literature Review

As mentioned previously, turbomachinery flow fields are inherently unsteady. The flow unsteadiness can have significant effects on the aerodynamic, aeroelastic, and aeroacoustic performance of turbomachines. The research in this field can be divided into three major categories, which inevitably can be intertwined. They are: linearized analysis, which addresses mainly aerodynamic issues related to aeroelastic and aeroacoustic applications; passage-averaged aerodynamic analysis, which models the influence of the upstream and downstream blade rows for an embedded blade row; and time accurate analysis for rotor-stator interaction.

1.3.1 Linearized Analysis of Unsteady Flows

The linearized theory for two-dimensional subsonic and supersonic flows, with the aid of a lot of empiricism, has been used for most current turbomachinery designs. Historically, the first attempts to model unsteady aerodynamics treated the flow as a small perturbation superimposed on a uniform flow field (Whitehead, 1960). The airfoil was considered a flat plate with zero incidence, although a circumferential jump in surface velocity and pressure was permitted. Then, thin-airfoil theory was used to calculate the unsteady response on the blade surface generated by the prescribed gusts, which were “frozen” and contained only the transverse component. Subsonic LINSUB (Whitehead,
1970) and supersonic LINSUP (Nagashima and Whitehead, 1977) programs were developed along this line. These programs are extremely fast and provide a useful tool in aeroelastic and aeroacoustic design. Since then, the unsteady linearized analyses were expanded to include axial gusts and interactive gusts. Still, the basic assumption for the linearized theory is that the unsteady flow is small in magnitude and is computed after a baseline steady state flow is established by other methods. The unsteadiness was also assumed to be harmonic in time and periodic over blade pitch. An extensive review on this subject is given in Verdon (1993).

The LINFLO (Hall and Verdon, 1991) and FINSUP (Whitehead, 1990) programs used potential flow as the baseline flow and allowed the interaction between gusts and the flow field. Euler solvers were used for calculation of the inviscid unsteady flows. One order of magnitude increase in computation time was needed compared to those of LIN-SUB and LINSUP. Shock waves were captured in the baseline flow and fitted into the unsteady calculation with a local shock-jump condition.

The above-mentioned methods treat the blade row as an isolated one. A model taking into account the neighboring blade rows was proposed by Hall and Silkowski (1997). It uses a small number of additional modes, which account for the influence of other blade rows through a set of reflection and transmission coefficients.

1.3.2 Passage-Averaged Aerodynamic Analysis

The model of passage-averaging lies within the theoretical framework established by Adamczyk (1985). It tries to quantify the steady state influence of the other blade rows’
unsteady flow on a blade row that is embedded in a multistage turbomachine. There is no need to apply this kind of model for an isolated blade row or a single stage turbomachine.

For a stator blade row in a multistage turbomachine, its time-averaged flow field has a circumferential periodicity not equal to its own blade pitch, if there is another stator with a different blade count present in the machine. This apparent discrepancy leads to the introduction of a so-called “passage-average” operator that is applied to the time-averaged flow field and governing equations. In the new set of passage-averaged equations, additional terms, which are derived from the interactions between the nondeterministic flow, the unsteady deterministic flow, and the passage-averaged flow, are present. These terms are similar to the Reynolds stress terms generated after applying the ensemble-averaging operator to the Navier-Stokes equation. Models are needed to relate these additional terms to the averaged flow field (Adamcyz, 2000); just as turbulence models are needed for the Reynolds stress terms.

The passage-averaged models are used in many simulations, for example, Hohn and Heinig (2000), Rhie, et al. (1998), Valkov and Tan (1999). Particular attention is paid to the spanwise and circumferential redistributions of total enthalpy and total momentum, wake recovery, and blade indexing. In Valkov and Tan (1999), flow through an axial multistage compressor is simulated. It is found that the stator efficiency is increased compared to the case where the upstream rotor wake is mixed out prior to entering the stator passage. Excitations due to tip vortex of the rotor on the stator are also investigated. It is concluded that the tip vortex acts in the same manner as a wake. The defects of total velocity and
total pressure across the tip vortex are the most important in representing the excitations due to the tip vortex.

A method similar to the passage-average models is the “mixing plane” approach. Across the interface between the blade rows, one of several possible circumferential averaging strategies is employed to pass information from the one blade row to another in the appropriate direction. A Navier-Stokes solver is used in computing the steady state flow through a three and half stage compressor in Dawes (1992), where a simple area-averaging scheme is used at the mixing plane. Although the mass is conserved across the plane, total pressure and enthalpy are not. A similar strategy is used in a later solver (Dawes, 1995).

In Edmunds, et al. (1999), the mixing plane is treated in a homogenous “mixing out” manner to obtain an averaged flow for passing on to the neighboring blade row. The averaging process is similar to let the vortical and potential disturbances dissipate over a long distance till the flow reaches a homogeneous state. Obviously, growth in dissipation and generation of entropy are involved in the process and these leads to additional errors for the solver.

1.3.3 Time Accurate Analysis for Rotor-Stator Interaction

With the advancement in computer power and numerical techniques, prediction methods based on more general physical models have been made possible. Time-accurate computation of unsteady flows certainly has higher fidelity than the previously discussed linearized theory and passage-average models. It accounts for the nonlinear effects and gives a more complete and realistic picture of the flow physics.
The first such computations is naturally the Euler unsteady computation by presuming the flow to be inviscid. A good deal of nonlinear effects is included (Giles, 1988; Giles, 1990). Some popular unsteady codes, most of which are based on potential or Euler analysis, are tested and compared by Manwaring and Wisler (1993) on benchmark cases for both turbine and compressor flows. The gust responses are Fourier decomposed and the contents of the harmonics are compared. Adequate prediction of the magnitude of the first harmonic of the unsteady pressure on the blade surfaces by the codes to various degrees. It is concluded the potential and Euler analyses can serve an important role in design applications.

One difficulty for the unsteady simulation is to account for the different blade count of different blade rows. Such difficulty arises from the large CPU and memory requirement to compute a large number of blade passages to retain periodicity, or in the case of only computing one or a smaller number of blade passages, the loss of periodicity over the pitch used. A number of techniques are introduced to resolve the latter problem. For example, a time-lagged approach is used by Erdos, et al. (1977) and a “time-tilting” method is implemented by Giles (1988). These methods allow for the computation be done in only one blade passage. Therefore, computing time and memory is reduced. But with certain assumptions being made about the nature of the unsteadiness, their range of application within which errors are reasonably small is limited.

With the trend toward high loading, closer spacing between rotor and stator in the design, the unsteadiness in the flow field becomes more and more severe. The interaction between various sources of the unsteadiness makes it imperative to understand not only
the unsteady pressure, but also the unsteady velocity field, unsteady blade boundary layer and unsteady vortices. One of the major problems with the Euler computation is that the code does not allow the physical decay of the upstream wake, thus overpredicting the unsteady response. Better understanding of the unsteady flow structure should be achieved through the use of the Navier-Stokes equations. Viscous solvers have become more and more popular. The unsteadiness is simulated either through prescribed unsteady boundary conditions (i.e., wake-passing) or a direct rotor/stator interaction. The progress in the development of the unsteady solvers has been toward solving multi-block and multi-passage and there-dimensional cases, i.e. Dorney and Davis (1993), Dorney, et al. (1995), Dorney and Schwab (1996), Dorney and Sharma (1997).

Two-dimensional unsteady rotor-stator interaction using the Thin-Layer Navier-Stokes (TLNS) equations was first computed for a single blade passage by Rai (1987). The method is extended to multi-passage to account for uneven blade count (Rai and Madavan, 1990). Three-dimensional computations are carried out for unsteady rotor-stator interaction for single-passage (Rai, 1989), and multi-passage (Madavan and Rai, 1993). An O-grid is used around the blade and a H-grid is built for the rest of the flow field. The O-grid and H-grid are overlaid where the two meet. At the interface, the H-grid blocks are patched together. Interpolation is needed to exchange information across the interface.

In Dorney, et al. (1995), the interaction between the blade rows is assumed to weak, and loosely coupled blade rows are computed. The interface between blade rows is interpolated over overlaid grid blocks. In Arnone and Pacciani (1995), phantom grid cells are extended to the neighboring blade row at the interface between blade rows. Linear
interpolation is used for the exchange of information. It is not conservative, but good accuracy is claimed based on comparison with measurements.

In Dorney and Sharma (1997), the three-dimensional unsteady flow field through a transonic compressor stage is calculated using four codes, which represent four classes of computation methods. Namely, the Fully Coupled Blade Row (FCBR) method, which is due to Rai (1989) for unsteady rotor-stator interaction; the Steady Coupled Blade Row (SCBR) method, which uses either a “mixing-out” concept (Dawes, 1992), or the passage-averaged models due to Adamczyk (1985) and Rhie, et al. (1998); the Steady Single Blade Row (SSBR) method that treat the blade row as an isolated airfoil; and the Loosely Coupled Blade Row (LCBR) method, due to Dorney, et al. (1995) mentioned above. Through the comparison of the computational results and available data, it is concluded that modeling the effects of the interaction between the rotor shock and the vane is necessary to get correct loss coefficients and efficiency. The results due to the SCBR and LCBR methods are reasonable, but that due to SSBR has significantly underpredicted loss coefficient. In Venable, et al. (1999) and Busby, et al. (1999), time-averaged and time-accurate results were obtained using several unsteady codes. Particularly, The effects of rotor-stator axial gap were studied.

Unsteady quasi-three-dimensional flow due to blade vibration was calculated by He (1993). Subsequently the full three-dimensional solutions were obtained (He and Denton, 1994). The TLNS with an algebraic turbulence model was used. A multigrid technique was developed by Arnone and Pacciani (1995) to accelerate their 2d unsteady code,
which solves the rotor and stator together and has the ability to handle multi-passage configurations.

Most of the computations mentioned above use the time-marching method for compressible flows. There are several problems with these compressible codes when applied to low Mach number flows. The main problem is that the algebraic matrixes become very stiff and convergence rates deteriorate significantly. The CFL time step limit becomes very restrictive for time-marching techniques, hence large number of time steps is needed to resolve an unsteady period and the CPU time required for a prediction is quite large. Also, there is large interaction between the long-wavelength errors and the in/out-flow boundaries when the Mach number goes to zero for a compressible code (Walker and Dawes, 1994). To overcome this problem, the artificial compressibility concept of Chorin (1967), is employed in the time-marching technique for many low-speed predictions (i.e. Sheng, et al., 1995). But the selection of the artificial sonic speed is somewhat arbitrary.

The pressure-based method can be an efficient alternative in this regard. For incompressible flows, the time step constraints are usually much more relaxed for the pressure-based method (Ho, 1995). In this research, both codes are based on the pressure-based method. They are developed for the prediction of three-dimensional unsteady incompressible flows, including rotor-stator interaction. A wake-passing strategy and full rotor-stator interaction capability are developed in the codes. These are at the top of the hierarchy of unsteady flow computations.

Validation of unsteady codes is very important in ensuring an accurate computation. Most of the works mentioned above compare only the unsteady pressure on the blade
Important viscous effects in unsteady flows are not addressed adequately. This can be achieved through the comparison of the unsteady velocity profiles, the secondary flows, and the boundary layer development. Accurate predictions of these viscous effects can be achieved through the use of the full Reynolds-averaged Navier-Stokes equations in the codes used in this research.

1.3.4 Grids Used in Turbomachinery Flow Computations

Traditionally, the unsteady flow is calculated on a single blade row. The unsteadiness is represented approximately by boundary conditions like the inlet transverse gust or the exit pressure wave. The real interactions between the adjacent blade rows are approximated. Solving two or more blade rows simultaneously is the natural choice to overcome this problem. Usually this requires solving flows in both absolute and relative frames simultaneously. Multi-block grid system for structured grids has to be used in order to make sure the grid for each blade row is stationary relative to the blade itself. Many examples of computations based on multi-block approaches can be found in literature, e.g. Moon and Liou (1989), Wright and Shyy (1993). An extensive review on this topic is provided in Shyy (1994).

The most difficult part of the multi-block approach is the transfer of the information between blocks. Certain criteria such as conservation of flow properties, stability, and accuracy have to be satisfied on the interface between the blocks. Two types of grid system are commonly used: patched grids and overlaid grids. Patched grids are separated into zones by the so-called patched boundaries. The zones have common boundaries. Overlaid
grids extend one grid zone into another grid zone and no common boundary between the zones is found.

Overlaid grids are commonly used in the interface of different types of grid, such as from C-grid or O-grid to H-grid, e.g., Dorney, et al. (1995), Rai (1987). They have the flexibility of distributing the grid points on the boundaries of different zones. But the transformation of information between zones is usually more difficult than the patched grid system. On the other hand, the patched grids are mostly used between different blade rows. In a typical rotor-stator configuration, the grids of the stator and the rotor are usually patched at a common axial plane located between the blade rows. The exchange of information between the zones is performed on a space that has one less dimension than the original problem. That is, for a two-dimensional computation, the patched boundary will be a line (one-dimensional); for a three-dimensional computation, the patched boundary will be a plane (two-dimensional). This reduction in dimension proves to be especially useful in the case where the rotor and stator are moving relative to each other in the computation. Much more work would be needed if overlaid grids were used in such a case.

In recent years, unstructured grid techniques have evolved and are becoming more widely employed for many CFD applications, e.g., Currie and Carsdoller (1998), Dawes (1995), Delanaye and Essers (1997), Sayma, et al. (2000). There is usually no need for grid overlapping when unstructured grids are used. In Sayma, et al. (2000), hybrid structured and unstructured grids are used for a three-dimensional unsteady Navier-Stokes solver. Information is linearly interpolated across the interface between the blade rows.
The most obvious advantage for the unstructured grid over the structured grid is that the former is the most suitable for the complex geometry commonly encountered in the turbomachinery configurations. Less grid distortion is possible compared to a structured grid, especially near blade leading and trailing edges. Also, grid adaptation is much easier to implement for the unstructured grid. Fine grids can be placed in regions where a high gradient of flow variables is present, such as in the regions of a wake, rotor tip gap, and shock wave.

The disadvantages for the unstructured grid system are that coding is more difficult because of the indirect addressing required; the grid is generally irregular and the numerical solution is sensitive to the grid quality; and accurate numerical schemes are more difficult to design. Also, generally more memory is required per grid element for the grids than those for structured grids. Though, as illustrated in this thesis, unstructured grids offer the possibility of needing fewer elements to resolve an upstream vortical disturbance in a blade passage.

Nonetheless, unstructured grids are still very attractive in the computation of turbomachinery flows. Particularly, as will be shown in this thesis, an unstructured grid allows efficient and accurate capture of wakes and transfer of information between blade rows in different frames of reference.

1.3.5 Multiphase Flows; Parallel Computing

Multi-phase (i.e., bubbly) flow is of relevance to surface ship propulsion applications. In particular, the bubbles that are generated along the ship hull free surface boundary are transported along and underneath the hull. Much of this bubble field can pass near
and/or though the propulsor. Accordingly, the bubbly wake of the ship, which is of significant interest in wake acoustic signature analysis, is significantly influenced by bubble-propulsor interactions.

With advances in computer architecture and power, lower cost of computer chips, and the development of relevant software, parallel computing has become more widely employed. It is very attractive for turbomachinery CFD applications, especially for computations involve multiple blade rows and passages.

1.4 Outline of Thesis

The rest of the thesis is organized in the following manner: first, the flow equations, discretization schemes, and numerical procedures are presented in Chapter 2. The common points and differences between the two codes adapted are listed. Then, computational results are given in the next chapters for four test cases. The first code is used in Chapters 3 and 4, while the second code is used in Chapter 5 and 6.

In Chapter 3, a simulation study is carried out for a compressor stator. Wake passing strategy is employed. The effects of the axial between the stator and the upstream rotor are investigated by varying the axial distance between the fixed incoming wake profile and the stator blade. Then, the effects of blade count ratio are studied with different number of inlet passing wakes in one stator pitch.

The three-dimensional steady state flow field through the inlet guide vane of a pump is computed in Chapter 4. Detailed comparison of predicted and measured wakes and secondary flows are made.
Full rotor-stator interaction simulations are carried out for a turbine stage in Chapter 5 and for the pump (studied in Chapter 3) in Chapter 6. An additional two-fluid computation demonstrates the solver’s ability in predicting multiphase flow fields.

Finally, conclusions based on these numerical studies are drawn. Recommendations for future work are also listed.
CHAPTER 2 GOVERNING EQUATIONS AND NUMERICAL PROCEDURES

2.1 The Governing Equations

The equations governing incompressible fluid flows are derived from the laws of conservation of mass and momentum. For flows through turbomachinery blade rows, they can be written in integral form in a rotational coordinate system as:

\[ \int_{V} \mathbf{W} \cdot dA = 0 \]  \hspace{1cm} (2.1)

\[ \rho \frac{\partial}{\partial t} \int_{V} \mathbf{W} dV + \rho \int_{A} \mathbf{WW} \cdot dA = \rho \int_{V} (F - 2\omega \times \mathbf{W} - \omega \times \omega \times \mathbf{W}) dV + \int_{A} [\tau] \cdot dA \]

or in differential form as:

\[ \nabla \cdot \mathbf{W} = 0 \]  \hspace{1cm} (2.2)

\[ \rho \frac{D\mathbf{W}}{Dt} = -2\rho \omega \times \mathbf{W} - 2\rho \omega \times \omega \times \mathbf{r} + \rho \mathbf{F} - \nabla p + \nabla \cdot [\tau] \]

where \( \rho \) is the density, \( \mathbf{W} = \mathbf{V} - \omega \times \mathbf{r} \) is the relative velocity, \( \mathbf{V} \) is the absolute velocity vector, \( V \) is the control volume, \( A \) is the surface of the control volume, \( \omega \) is the blade rotation vector, \( \mathbf{r} \) is the radius vector, \( \mathbf{F} \) is the body force, \( p \) is the pressure, and \( [\tau] \) is the stress tensor. The first two terms on the right hand side of the second equation in Eq. 2.2 are the Coriolis force and the centrifugal force, respectively. For flows through a stationary blade row, \( \omega \) is zero, and \( \mathbf{W} = \mathbf{V} \). The Navier-Stokes equations shown above can be written in the Cartesian coordinate system in Reynolds-averaged form for three-dimensional, unsteady flows:
where $u_i$ are the mean velocity components, $x_i$ are the independent variables, $F_i$ is the body force component, $\mu$ is the molecular viscosity, $\overline{u'_i u'_j}$ are the Reynolds stress tensor components, $S$ is the source term that contains the Coriolis force and the centrifugal force, and $i = 1, 2, 3$ represents components in $x$, $y$, $z$-directions, respectively.

For three-dimensional flows, there are four equations and four unknowns, $u$, $v$, $w$, and $p$, not counting the Reynolds stress, which needs additional equations to close the problem. In this research, the $k$-$\varepsilon$ model is used for the turbulence closure. The Boussinesq approximation, which assumes that the Reynolds stress is proportional to the gradient of the mean velocity, is adopted as

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

(2.4)

where $\mu_t$ is the turbulent eddy viscosity, $k = \frac{1}{2} \overline{u'_i u'_i}$ is the turbulent kinetic energy and $\varepsilon = \frac{\partial u'_i}{\partial x_j} \frac{\partial u'_i}{\partial x_j}$ is the turbulent energy dissipation rate. The equations for the turbulent kinetic energy and turbulent dissipation rate including the low-Reynolds-number effects are as follows:
where the turbulent eddy viscosity is expressed as: \( \mu_t = f_\mu C_\mu \rho k^2 / \varepsilon \). The production term is \( P_k = \frac{\partial u_i}{\partial x_j} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \), the constants have values of \( C_\mu = 0.09, \sigma_\varepsilon = 1.3 \); and for the high-Reynolds-number model due to Jones and Launder (1972), \( C_1 = 1.55, C_2 = 2, f_2 = 1 \); for the low-Reynolds-number model due to Chien (1982), \( C_1 = 1.35, C_2 = 1.80, \) and

\[
\begin{align*}
\rho \frac{\partial k}{\partial t} + \rho \frac{\partial}{\partial x_i} u_i k &= \frac{\partial}{\partial x_i} \left[ \left( \mu + \mu_t \right) \frac{\partial k}{\partial x_i} \right] + P_k - \rho \varepsilon \\
\rho \frac{\partial \varepsilon}{\partial t} + \rho \frac{\partial}{\partial x_i} u_i \varepsilon &= \frac{\partial}{\partial x_i} \left[ \left( \mu + \mu_t \right) \frac{\partial \varepsilon}{\partial x_i} \right] + c_1 \varepsilon P_k - c_2 f_2 \rho \frac{\varepsilon^2}{k}
\end{align*}
\]

(2.5)

(2.6)

The complete set of governing equations (for velocity components, \( k \), and \( \varepsilon \)) can be written in terms of a single general convection-diffusion equation for an arbitrary scalar dependent variable, \( \phi \), in integral and differential forms as:

\[
\rho \frac{\partial}{\partial t} \int_V \phi dV + \rho \int_V \nabla \phi \cdot \mathbf{A} dA = \int_V S_\phi dV + \int_\Gamma [\tau] \cdot \mathbf{A} dA
\]

(2.8)

\[
\rho \frac{\partial \phi}{\partial t} + \rho \frac{\partial u_i \phi}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \Gamma_\phi \frac{\partial \phi}{\partial x_j} \right) + S_\phi
\]

(2.9)

where \( \Gamma_\phi \) is the effective diffusion coefficient and \( S_\phi \) denotes the source term. This formulation greatly simplifies the coding and debugging efforts.
As mentioned in the introduction, there are two codes adapted in this research. For the sake of convenience, the first code will be referred to as code A and the second code B. The common features and differences between them in terms of discretization, grid system used, solution procedure, etc., will be discussed. Then, a summary of the structures of the two codes is presented.

To accommodate the structured grid system used in code A for complex geometries, the generalized body-fitted coordinates, $\xi, \eta, \zeta$, are used for Eq. 2.9 as:

$$
J \frac{\partial \rho \phi}{\partial t} + \frac{\partial}{\partial \xi} (\rho J G_1 \phi) + \frac{\partial}{\partial \eta} (\rho J G_2 \phi) + \frac{\partial}{\partial \zeta} (\rho J G_3 \phi)
$$

$$
= \frac{\partial}{\partial \xi} \left[ \Gamma_\phi J (\alpha \phi_\xi + \theta_1 \phi_\eta + \theta_2 \phi_\zeta) \right] + \frac{\partial}{\partial \eta} \left[ \Gamma_\phi J (\theta_1 \phi_\xi + \beta \phi_\eta + \theta_3 \phi_\zeta) \right] + \frac{\partial}{\partial \zeta} \left[ \Gamma_\phi J (\theta_2 \phi_\xi + \gamma \phi_\eta + \phi_\zeta) \right]
$$

\text{(2.10)}

where the previously undefined variables are defined in the nomenclature. The pressure is solved through the continuity equation and its formula will be presented later.

### 2.2 Data Structure

For code A, since single-block structured H-grids are used, the data structure is straightforward. All flow variables are stored on the grid nodes, which are where the grid lines intersect. Grid node is also referred to as grid vertex in this thesis. Volume grid is referred to as grid cell or grid element. As shown in Figure 2.1 for the computation domain, a grid node P has six neighboring nodes in three dimensions. The box shown is not a grid cell, but rather the control volume for node P. Surfaces of the control volume are located half way between respective set of nodes.
For code B, since a fully unstructured grid system is used, the data structure is more complex. The code can handle any arbitrary polyhedral grid cell, as shown in Figure 2.2. Here, the surfaces bounding the cell are called faces. Any of the actual grid point shown is called a vertex (or node). Grid lines connecting the vertices are called edges.

Code B uses a face-based data structured to organize the grids. Primary flow variables such as velocity, pressure, and turbulence quantity are stored on grid cell centers. The convection velocity is stored on grid faces. The most fundamental data structure is the so-called “fedge”, which means “face edge”, or “edge of a face.” All the connectivity between hierarchy of the grid starts from the fedge. For example, a fedge has a pointer that points to an edge whose numbering is global. Then, an edge has two pointers that point to the vertices on either end of the edge. A fedge also points to the face the fedge belongs to. Finally, each face has two bounding cells. The data structure hierarchy is also shown in Figure 2.2.

There is a distinction between the “internal” faces and fedges and the boundary faces and fedges. The latter is discussed separately in more detail later. By definition, cells are all internal. Faces on which no boundary conditions are needed are also internal. So are their bounding fedges. Since each edge belongs to at least one internal face, even for those edges that lie on a boundary face, all edges are internal. Therefore, all vertices are internal.

One-dimensional arrays are assigned to internal vertices, edges, fedges, faces, and elements. Physical Cartesian coordinates for vertices are stored in such arrays. Coordinates for faces and elements are also stored. They are calculated from the coordinates of
the vertices. Primary flow variables such as pressure, velocity components, k, and ε are stored on the elements only. Convection velocity is stored on the internal faces.

2.3 Discretization Schemes

2.3.1 Discretization Schemes for General Transport Equations in Code A

A control volume approach is used in the discretization of governing equations (Eq. 2.10). In order to represent the convection term with a second-order accuracy scheme, the convection term is discretized with a central differencing scheme. However, the central difference scheme is unstable when solely used for the convection term. The main reason is that the central differencing is not dissipative and the physical viscous terms can not damp errors with high wave numbers, even in viscous flow computations with small grid spacing (Kunz, et al., 1992). The central difference scheme also introduces the so called “checkerboard effect” (Patankar, 1980), or uncoupled odd-even grid points. All the high order upwinding schemes used in the time marching methods are more accurate than the basic upwind scheme, but they inherently include certain amount of artificial dissipation (Pulliam, 1986). Also, it is shown in Basson and Lakshminarayana (1994) that the more accurate QUICK scheme (Leonard, 1979) is equivalent to central differencing plus third-order and fourth-order dissipation terms. Thus, the disadvantage of these higher order upwind schemes is the lack of total control on the amount of numerical dissipation. In view of this, the scheme developed in Basson and Lakshminarayana (1994), with a second-order and a fourth-order artificial dissipation added explicitly, is used to control the numerical dissipation.
Because of the non-regular geometries encountered in most practical applications, staggered grids become inconvenient and less efficient. Therefore, a non-staggered grid system is implemented, i.e., all the variables are located at the grid points while the coefficients are calculated at the control surfaces around the grid points. To obtain the coefficients at a control surface, simple linear interpolation is used on the transformation plane.

The difference approximation of convective terms can be demonstrated by the following procedure (see Figure 2.1):

\[
\frac{\partial}{\partial \xi} (\rho J G_1 \phi) = (\rho J G_1 \phi)_e = (\rho J G_1)_e \phi_e - (\rho J G_1)_w \phi_w 
\]  

(2.11)

where a simple linear interpolation is used at the control surfaces. That leads to

\[
\phi_e = \frac{1}{2} (\phi_E + \phi_P) \quad \text{and} \quad \phi_w = \frac{1}{2} (\phi_P + \phi_W).
\]

Substituting these into Eq. 2.11, the final form for the convection term can be written as:

\[
\frac{\partial}{\partial \xi} (\rho J G_1 \phi) = \frac{1}{2} (\rho J G_1)_e \phi_E + \frac{1}{2} (\rho J G_1)_e \phi_p - \frac{1}{2} (\rho J G_1)_w \phi_p - \frac{1}{2} (\rho J G_1)_w \phi_w 
\]  

(2.12)

Similar expressions can be derived for the other two convection terms. So are the diffusion terms differenced:

\[
\frac{\partial}{\partial \xi} [\Gamma \phi (\alpha \phi_\xi + \theta_1 \phi_\eta + \theta_2 \phi_\zeta)] = [\Gamma \phi (\alpha \phi_\xi + \theta_1 \phi_\eta + \theta_2 \phi_\zeta)]_w 
\]  

(2.13)

\[
= (\Gamma \phi J \alpha)_e (\phi_\xi)_e - (\Gamma \phi J \alpha)_w (\phi_\xi)_w + (\Gamma \phi J \theta_1)_e (\phi_\eta)_e - (\Gamma \phi J \theta_1)_w (\phi_\eta)_w \\
+ (\Gamma \phi J \theta_2)_e (\phi_\zeta)_e - (\Gamma \phi J \theta_2)_w (\phi_\zeta)_w
\]

where the derivatives at the control faces are \((\phi_\xi)_e = \phi_E - \phi_P\) and \((\phi_\xi)_w = \phi_P - \phi_W\).

The cross-derivative terms are:
In the present formulation, an artificial dissipation scheme developed by Basson and Lakshminarayana (1994) is employed to maintain stability of the numerical algorithm. An explicit amount of second and fourth order artificial dissipation terms can be added to the convection-diffusion equation in the following respective forms:

\[
(\phi_\eta)_e = \frac{1}{4}(\phi_N - \phi_S + \phi_{NE} - \phi_{SE}) \\
(\phi_\eta)_w = \frac{1}{4}(\phi_N - \phi_S + \phi_{NW} - \phi_{SW}) \\
(\phi_\zeta)_e = \frac{1}{4}(\phi_T - \phi_B + \phi_{TE} - \phi_{BE}) \\
(\phi_\zeta)_w = \frac{1}{4}(\phi_T - \phi_B + \phi_{TW} - \phi_{BW})
\]

These terms are included to provide additional stability to the algorithm. The leading coefficients, 1/2 and 1/8, of the second-order and the fourth-order numerical dissipation are chosen to be consistent with the artificial dissipation scheme of the traditional time-marching techniques. The artificial dissipation coefficients, \(\epsilon_2\) and \(\epsilon_4\), are case-dependent and user-specified. The central difference scheme is also used for discretization of the dissipation terms. The discretization of the fourth order dissipation results in a six-point formulation and a sparse matrix. In order to use the tridiagonal matrix solving technique (ADI), the fourth order dissipation is treated partially-implicitly. It can be demonstrated by differencing the following terms:
For temporal discretization, a three-point backward differencing scheme is selected. Thus, second order accuracy in time domain is preserved. The unsteady term in Eq. 2.10 can be written as:

\[
\frac{\partial}{\partial t} \rho \phi = J \rho \frac{3 \phi^{n+1}_P - 4 \phi^n_P + \phi^{n-1}_P}{2\Delta t}
\] (2.17)

where \(\Delta t\) is the time step and superscripts \(n + 1\), \(n\), and \(n - 1\) represent the next, current, and previous time level, respectively.

Using a similar process for the remaining terms in Eq. 2.10, the following difference approximation for the general convection-diffusion equation is obtained:

\[
A_p^* \phi_p = \sum_{nb} A_{nb} \phi_{nb} + S_{\phi}^*
\] (2.18)

where subscript \(nb\) stands for all the neighboring grid nodes, E, W, N, S, T, and B; and
2.3.2 Discretization Schemes for General Transport Equations in Code B

The control volume method is also used in code B. Considering Eq. 2.8. Inviscid
and viscous fluxes are accumulated by sweeping through internal and boundary faces. Each face contributes to its bounding cells and after the sweeping, the correct terms for each cell are obtained. The inviscid flux term is evaluated as:

\[ \int_{\Lambda} \phi \mathbf{W} \cdot d\mathbf{A} = \sum_{f} C_f \phi_f \]  \hspace{1cm} (2.21)

where \( C_f \) is the mass flux across face, \( f \), and \( \phi_f \) is the value of \( \phi \) evaluated at the face. The summation is taken over all faces bounding the element (see Figure 2.2). \( C_f \) is evaluated based on field variable prior to the solution of the transport equation for \( \phi_f \). So it is a lagged coefficient-linearization strategy similar to Clift and Forsyth (1994). Second order accuracy is obtained by evaluating \( C_f \) using a central plus fourth difference pressure artificial dissipation term due to Rhie and Chow (1983):

\[ C_f = \overline{\mathbf{W}_f} \cdot \mathbf{A}_f + B (\nabla \mathbf{p}_f \cdot \mathbf{A}_f - \Delta p |\mathbf{A}_f|^2) \]  \hspace{1cm} (2.22)

and by evaluating \( \phi_f \) (Lien, 2000), as:

\[ \phi_f = \phi_U + (\nabla \phi \cdot d\mathbf{r})_U \]  \hspace{1cm} (2.23)

where subscript \( U \) designates the quantity associated with the element upwind of face, \( f \), and \( d\mathbf{r} \) is the vector from the upwind cell center to the face center. The overbar denotes a geometrically weighted mean at the face, i.e., referring to Figure 2.3:
\[
\tilde{\phi} = (1 - s)\phi_1 + s\phi_2 \\
s \equiv \frac{\delta s_1}{\delta s_1 + \delta s_2}
\]  (2.24)

and \( \Delta \) designates a difference across the face (i.e., \( \Delta \phi \equiv \phi_2 - \phi_1 \)). The first order contribution in Eq. 2.23, \( \phi_U \), is treated implicitly. The second order term is treated explicitly.

Dissipation parameter \( B \) in Eq. 2.22 is scaled as:

\[
B \equiv \frac{V}{A_p}
\]  (2.25)

where \( A_p \) is the momentum equation point influence coefficient arising from convection, diffusion and implicit sources.

Neglecting cross-diffusion and dilatation, the viscous flux in the momentum equations (last term in Eq. 2.8) can be written at an element face as:

\[
\int \mu_e \nabla \mathbf{W} \cdot d\mathbf{A}
\]  (2.26)

where \( \mu_e = \mu + \mu_t \) is the effective viscosity. Referring to Figure 2.3, the gradient of a scalar, \( \phi \), on the face can be written as:

\[
\nabla \phi = \nabla \phi - \left( \nabla \phi \cdot \frac{s_{12}}{|s_{12}|} \right) \mathbf{e}_{12} + \left( \nabla \phi \cdot \frac{s_{12}}{|s_{12}|} \right) \mathbf{e}_{12} \quad (2.27)
\]

where \( \mathbf{e}_{12} \) is the unit vector from \( n_1 \) to \( n_2 \), the terms labeled \( A \) represent components of the gradient that are orthogonal to \( \vec{s}_{12} \). These terms are generally small (for hexahedral or prismatic elements extruded from geometric surfaces, neglecting them is nearly equivalent
to the thin-layer assumption). Their discrete form is treated explicitly in the solution of the
momentum equations (term S in Eq. 2.50 below). The terms labeled B represent compo-
nents of the gradient that are parallel to $\mathbf{s}_{12}$. These are discretized as:

\[
\int_{f} \mu_{e} \nabla \mathbf{W} \cdot d\mathbf{A} = \frac{(\mu_{e})_{f}}{d\xi} \left( \nabla \mathbf{W} \cdot \frac{d\xi}{|d\xi|} \right) (\mathbf{e} \cdot d\mathbf{A})
\]

\[
= \frac{(\mu_{e})_{f}}{d\xi} (\mathbf{W}_{2} - \mathbf{W}_{1}) (d\xi \cdot d\mathbf{A})
\]

and are treated implicitly (terms $A_{p}$ and $A_{nb}$ in Eq. 2.50 below).

Gradients that appear in the flux calculations, and elsewhere, are computed using
Gauss’ Law:

\[
\nabla \phi = \frac{1}{f} \sum_{f} \phi_{f} A_{f}
\]

(2.29)

with internal face values of $\phi_{f}$ computed from Eq. 2.24, and the summation taken over all
faces bounding an element. Eq. 2.29 is computed by sweeping all internal and boundary
faces, accumulating adjacent element contributions to $V$ and $\phi_{f} A_{f}$ from the face.

2.3.3 Pressure Equation for Code A

It is clear from the governing equations that pressure is indirectly coupled with
velocity components. An equation for pressure can be derived by taking the cross deriv-
atives of the momentum equations and applying the continuity equation to obtain a Poisson
equation. A second option is to discretize the momentum equations first and then substi-
tute them into the continuity equation to express the pressure in terms of velocity components. The second approach is chosen for code A.

The convection across the control surfaces must be known to formulate the continuity equation. Although a non-staggered grid is more suitable for the generalized coordinates, it suffers from the pressure oscillation when the convection along the control surfaces is calculated from the nodal values. In order to overcome this problem, Rhie and Chow (1983) employed a pressure-weighting scheme. It is used to link the 2Δx scheme with Δx pressure variation. It has been proven that this scheme is equivalent to adding a fourth-order artificial dissipation to the pressure equation (Basson and Lakshminarayana, 1994). In the present work, a fourth-order numerical dissipation scheme is used. This is sufficient to provide a smooth pressure field.

The standard pressure correction scheme is used to set up the coefficients of the pressure equation. In order to ensure the consistency between the pressure equation and the discrete form of the momentum equations, the derivation therefore uses the difference form of the momentum equations. For the u-momentum equation, its finite difference form can be expressed as follows:

\[ A^p u_p = \sum_{nb} A^{nb} u_{nb} + S_u - J(\xi_x p_{\xi} + \eta_x p_{\eta} + \zeta_x p_{\zeta}) \]  

(2.30)

where \( nb = e, w, n, s, t, b \), \( S_u \) is the source term of the u-momentum equation excluding the pressure derivatives. Eq. 2.30 can be re-written as the following equation:
where

\[ E = \frac{1}{a^p - \sum_{nb} a^{nb}} \]  

\[ \hat{u}_p = E \left( \sum_{nb} a^{nb} (u_{nb} - u_p) + S_u \right) \]

Employing a similar approach for the v- and w-momentum equations, expressions can be derived for v and w as:

\[ v_p = \hat{v}_p - EJ(\xi_x p_\xi + \eta_x p_\eta + \zeta_x p_\zeta) \]  

\[ w_p = \hat{w}_p - EJ(\xi_z p_\xi + \eta_z p_\eta + \zeta_z p_\zeta) \]

In order to simplify the derivations in the generalized coordinates system, new contravariant velocities are defined as:

\[ \hat{G}_1 = \xi_x \hat{u} + \xi_y \hat{v} + \xi_z \hat{w} \]

\[ \hat{G}_2 = \eta_x \hat{u} + \eta_y \hat{v} + \eta_z \hat{w} \]

\[ \hat{G}_3 = \zeta_x \hat{u} + \zeta_y \hat{v} + \zeta_z \hat{w} \]

The contravariant velocities can then be represented as the following form:

\[ \rho J\hat{G}_1 = \rho J\hat{G}_1 - \rho EJ^2 (\alpha p_\xi + \theta_1 p_\eta + \theta_2 p_\zeta) \]

\[ \rho J\hat{G}_2 = \rho J\hat{G}_2 - \rho EJ^2 (\theta_1 p_\xi + \beta p_\eta + \theta_3 p_\zeta) \]

\[ \rho J\hat{G}_3 = \rho J\hat{G}_3 - \rho EJ^2 (\theta_2 p_\xi + \theta_3 p_\eta + \gamma p_\zeta) \]

These equations are used to link the pressure equation with the velocity components. They will be substituted into the generalized continuity equation to derive the pres-
sure equation used in the present numerical scheme. If second-order central difference is
used, the continuity equation can be approximated as:

\[(\rho J G_1)_w^e + (\rho J G_2)_s^n + (\rho J G_3)_b^t = 0 \quad (2.36)\]

Substitution of the corresponding terms in Eq. 2.35 into Eq. 2.36, one can have the equa-
tion for pressure as follows:

\[
\begin{align*}
0 &= -(A^{cw} p_\xi)_w^e - (A^{ns} p_\eta)_w^e - (A^{tb} p_\zeta)_w^e + (\rho J G_1)w^e + (\rho J G_2)_s^n + (\rho J G_3)_b^t \\
&- [E_2J(\theta_1 p_\eta + \theta_2 p_\zeta)]w^e - [E_2J(\theta_1 p_\xi + \theta_3 p_\zeta)]s^n - [E_2J(\theta_2 p_\xi + \theta_3 p_\eta)]b^t
\end{align*} \quad (2.37)
\]

where

\[
\begin{align*}
A^{cw} &= \rho EJ^2 \alpha \\
A^{ns} &= \rho EJ^2 \beta \\
A^{tb} &= \rho EJ^2 \gamma \\
E_2 &= \rho EJ
\end{align*} \quad (2.38)
\]

Similar finite approximations for cross derivatives are used for the pressure deriv-
atives. However, cross derivatives involves the pressure will be kept in the same form and
are treated as source terms. The final difference form of the pressure equation is obtained:

\[
\begin{align*}
A^p p_p &= (A^{cw})_e p_E + (A^{cw})_w p_W + (A^{ns})_n p_N + (A^{ns})_s p_S + (A^{tb})_l p_T + (A^{tb})_b p_B \\
&- (\rho J G_1)_w^e - (\rho J G_2)_s^n - (\rho J G_3)_b^t \\
&+ [E_2J(\theta_1 p_\eta + \theta_2 p_\zeta)]w^e + [E_2J(\theta_1 p_\xi + \theta_3 p_\zeta)]s^n + [E_2J(\theta_2 p_\xi + \theta_3 p_\eta)]b^t
\end{align*} \quad (2.39)
\]

where
To obtain the equation for pressure corrections, a simple relation between the pressure and correct pressures is used. The pressure is assumed to be equal to the sum of its estimated (*) and correction (’) values. i.e., \( p = p^* + p' \), which is substituted into Eq. 2.39 and the contributions from \( u_p' \), \( v_p' \) and \( w_p' \) are neglected, the velocity equations are written as follows:

\[
\begin{align*}
\hat{u}_p &= \hat{u}'_p - EJ(\hat{\xi}_x p_\xi + \eta_x p_\eta + \zeta_x p_\zeta) \\
\hat{v}_p &= \hat{v}'_p - EJ(\hat{\xi}_y p_\xi + \eta_y p_\eta + \zeta_y p_\zeta) \\
\hat{w}_p &= \hat{w}'_p - EJ(\hat{\xi}_z p_\xi + \eta_z p_\eta + \zeta_z p_\zeta)
\end{align*}
\] (2.41)

The pressure correction equation reduces to the following form:

\[
\begin{align*}
A^p_{pp'} &= (A^{ew}_e)_{Pe} + (A^{ew}_w)_{Pw} + (A^{ns}_n)_{PN} + (A^{ns}_s)_{Ps} + (A^{tb}_t)_{P_T} + (A^{tb}_b)_{Pb} \\
&- (\rho JG_1^*)_w - (\rho JG_2^*)_s - (\rho JG_3^*)_b \\
&+ [E_2 J(\theta_1 p_\eta + \theta_2 p_\zeta')]_w + [E_2 J(\theta_1 p_\zeta + \theta_3 p_\eta')]_s + [E_2 J(\theta_2 p_\zeta' + \theta_3 p_\eta')]_b
\end{align*}
\] (2.42)

where the contributions from \( \hat{G}_1' \), \( \hat{G}_2' \), and \( \hat{G}_3' \) are again neglected.

The last three terms are the cross-derivative terms. To improve the convergence of the mass flow rate during each time step, the cross derivative terms of the discretized pressure equations are included and treated explicitly as source terms. The source terms of the discretized pressure equation consist of mass error (those terms in the second line of Eq. 2.42) and the cross derivative terms due to coordinate transformation. During the iteration procedure for solving discretized pressure equation, the cross derivative terms are updated after each iteration to reflect the pressure changes.
To prevent pressure oscillations due to the central differencing scheme, a fourth-order artificial dissipation terms similar to the one added to the general convection-diffusion scheme is employed in the pressure corrections equation. It is represented as follows:

\[
\frac{1}{4}\varepsilon_{pw}\left[\frac{\partial}{\partial \xi}\left(\frac{a}{\partial \xi^3}\right) + \frac{\partial}{\partial \eta}\left(\frac{a^{ns}}{\partial \eta^3}\right) + \frac{\partial}{\partial \zeta}\left(\frac{a^{nb}}{\partial \zeta^3}\right)\right] \quad (2.43)
\]

where \(\varepsilon_{pw}\) is the coefficient specified by users. When \(\varepsilon_{pw}\) is equal to 1, it corresponds to the original pressure-weighting scheme.

### 2.3.4 Pressure Equation for Code B

For incompressible flows, a SIMPLE-C (Van Doormaal and Raithby, 1984) based pressure-velocity corrector relation is applied to develop an elliptic pressure correction equation:

\[
A_p\Delta p - \sum_{nb} A_{nb}\Delta p_{nb} = -\dot{m}^* \quad (2.44)
\]

where \(A_p\) and \(A_{nb}\) are influence coefficients arising from the assumed pressure-velocity correction relation,

\[
\Delta u = -\frac{V}{A_p\partial x}\Delta p \quad (2.45)
\]

and \(\dot{m}^*\) is the net mass flux through the element based on the velocity field obtained after solution of the momentum equations. Pseudo-time stepping is not employed for the pressure corrector equation in order to achieve a measure of mass conservation at each
pseudo-time step, and, as result, the discrete pressure corrector equation system is symmetric positive semi-definite \((A_p = \sum A_{nb})\).

### 2.4 Boundary Conditions

There are usually four different types of boundary conditions encountered in the computation of turbomachinery flow fields. They are inlet, outlet, solid wall, and periodic boundaries. For two-dimensional flow computation, a stream sheet that contains one cell in the radial direction is used. Additional symmetry boundary condition is applied to the upper and lower (in the radial direction) faces.

In code A, these boundary conditions are applied directly to respective grid nodes. For example, the inlet boundary is used assigned with \(I = 1\) and the inlet boundary condition is specified at \(I = 1\). However for code B, since the nodes are randomly numbered in the grid file, grid boundaries have to be specified along with their specific type in the grid file. This is accomplished by listing the boundary faces and fedges for each boundary type. Similar pointers to cells and edges are also listed. All boundary conditions are then treated implicitly in the code in the formation of the influence coefficients for the transport scalars.

The velocity components are prescribed at the inlet for the momentum equations. The inlet turbulence kinetic energy and dissipation rate are either specified or determined by specifying an inlet turbulence intensity, \(T_u\), and the ratio of turbulent to laminar vis-
cosity factor, \( \mu_r \), associated with the inlet flow. This viscosity ratio controls the inlet turbulence length scale. The inlet turbulent kinetic energy is determined from:

\[
k_1 = \frac{3}{2} (T_u V_1)^2
\]  

where \( V_1 \) is the inlet total velocity and subscript 1 represents the inlet condition. The inlet dissipation rate is then determined from the following relation:

\[
\varepsilon_1 = \rho_1 C_{\mu} \frac{k_1^2}{\mu_r}
\]

At the exit, all variables in the scalar-transport equations are calculated directly from the convection-diffusion equations without using any assumptions. Therefore, the computation of all the derivatives uses a one-sided differencing approach and requires special coding at the outlet boundary.

The no-slip wall boundary condition is specified for \( u, v, w \) and \( k \) in the relative frame of reference on the solid walls. The boundary condition for \( \varepsilon \) depends on the turbulence model being used.

Specifically, the incorporation of the periodic boundary condition is different in the two codes. In code A, since flow variables are stored on the grid nodes, it would be reasonable to explicitly force each transport scalar to have identical values on each node on the lower periodic boundary and its respective node on the higher periodic boundary. (Here for the sake of convenience, lower or low represents the periodic boundary that has lower value of theta; and higher or high represents the periodic boundary that has lower value of theta.) For example,
where subscripts low and high represent conditions on opposite periodic boundaries and

\[ \phi = \frac{2\pi}{n} \] with \( n \) is the blade count. As can be seen from Eq. 2.49, the \( v \)- and \( w \)-momentum equations are coupled at the periodic boundaries. One reasonable alternative is to treat the cosine terms implicitly and to treat the sine terms explicitly. But, even though the angle \( \phi \) is small for typical blade numbers in turbomachinery blade rows, it is found the solutions converge very slowly if these sine terms are treated explicitly, especially for the pressure equation. Therefore, an additional grid point is introduced above the periodic points and the necessary coefficients at this grid point is calculated and stored. The periodic points are then solved implicitly.

For code B, the treatment of the periodic boundary condition is different. Since there are pointers used for faces and cells, it would be more convenient to do the following. The high periodic boundary faces are treated both as internal and boundary faces, while the low periodic faces do not exist in the data structure. The bounding cells for an internal periodic face would be a blade pitch apart in the circumferential direction. Special treatments for the geometrical parameters are needed here. Also, wall functions are used near solid surface boundaries for the \( k-\epsilon \) equations.
For incompressible flows the pressure appears only as gradient terms in the momentum equations. Thus, the absolute values of the pressure are not important. The other distinct feature of pressure is the boundary values of the pressure equation are unknown. Careful specification of the pressure boundary conditions is essential to the convergence to a correct pressure field.

Pressure is handled by the same approach as the velocities on the periodic plane. The normal derivative of the pressure is assumed zero at the solid surface, which is a reasonable approach for high Reynolds number flows. Furthermore, the Navier-Stokes computation usually requires dense grids near the solid surfaces; therefore, the distance between the first grid point and the surface is very small. The static pressure is specified at the exit, as is commonly used in most computational approaches.

The pressure is calculated by assuming that the streamwise gradient is zero at the inlet. Since Neumann boundary conditions are used, the pressure at one grid point must be chosen. This control point is taken to be the mid-channel point at the exit plane for turbo-machinery configurations. For three-dimensional computations, the pressure is assumed to be uniform circumferentially and the radial equilibrium equation is used to determine the spanwise pressure distributions.

2.5 Solution Procedure for Code B

In code A, the systems of all linear equations are solved using the ADI method. Here, more details of the solver in code B are presented.
2.5.1 Transport Scalars

Invoking a dual-time formulation, the discretized governing equations for transport scalar, \( \phi \) (velocity components and turbulence scalars), can be written in \( \Delta \)-form as:

\[
\begin{align*}
\left[ A_p + \frac{\rho V}{\Delta t} + \frac{3 \rho V}{2 \Delta t} \right] \Delta \phi_p - \sum_{nb} A_{nb} \Delta \phi_{nb} &= \sum_{nb} A_{nb} \phi_{nb}^{n+1,m} - A_p \phi_p^{n+1,m} + S - \\
&= \left[ \frac{3 \rho V}{2 \Delta t} \phi_p^{n+1,m} - \frac{2 \rho V}{\Delta t} \phi_p^n + \frac{\rho V}{2 \Delta t} \phi_p^{n-1} \right]
\end{align*}
\]

where \( A_p \) and \( A_{nb} \) are influence coefficients arising from convection, diffusion and implicit sources, and \( \Delta \phi \equiv \phi^{n+1,m+1} - \phi^{n+1,m \ 0.5 \leq \omega \leq 0.7} \) and the pseudo-time step is evaluated from:

\[
\Delta \tau = \frac{\omega}{1 - \omega} \frac{\rho V}{A_p}
\]

It has been observed in Venkateswaran, et al. (1997) that such a specification is equivalent to a local time stepping procedure that accommodates CFL and von Neuman stability. For physical transients, pseudo-time steps correspond to sub-iterations of the SIMPLE-C algorithm.

Equation 2.50 represents a coupled system of NN equations for the NN unknowns \( \phi \):
where coefficient matrix $A$ has the form:

$$A\phi = S$$

(2.52)

with

$$A = \begin{bmatrix} U & L & \vdots \\ P & \end{bmatrix}$$

(2.53)

$$P = A_p + \frac{\rho V}{\Delta t} + \frac{3\rho V}{2\Delta t}$$

(2.54)

For the diagonal dominance preserving discretizations employed, conventional iterative schemes will have diagonally dominant iteration matrices with spectral radii less than or equal to the underrelaxation factor, $\omega$ (Kunz, et al. 2001). Accordingly, here and elsewhere, e.g., Wright and Smith (2001), a simple point Jacobi scheme is consistently employed for solving Equation 2.50 for all scalars ($u_i, k, \varepsilon$); as this scheme is guaranteed to provide adequate convergence within several sweeps. For the momentum equations, all three velocity components are solved for all fields simultaneously using point Jacobi iteration.
2.5.2 Continuity Equation Linear Solution Strategy

An algebraic multigrid solver (AMG) is used for the solution of Eq. 2.44. The basic idea is to represent the error of the solution on successively coarser grids in order to eliminate the long wavelength components. For AMG, the representation on coarse grids is done by agglomerating fine grid nodes and manipulating the corresponding fine grid matrix coefficients and source terms (Raw, 1996). Coarse grid nodes are comprised of an integral number of fine grid cells. For each coarse grid, a correction term is computed such that the following condition is satisfied

\[ \sum R_i^{\text{fine}} = 0 \]  

where the summation involves all the fine grid node residuals that comprise a given coarse grid cell. The coarse grid correction is added to each of the associated fine grid nodes.

2.5.3 Coarsening Strategies

Two options are available to control the agglomeration/coarsening process: isotropic and directional. Isotropic coarsening is illustrated in Figure 2.4a. The shaded (fine grid) cell is simply grouped with its four neighbors, which are outlined in bold.

CFD calculations often employ stretched grids near walls, for example, to capture gradients in the flow field. Numerically, the high aspect ratio characteristic of these cells can lead to long wavelength error components in the stretching direction. To address this situation the AMG implementation includes a directional coarsening capability based on the cell facial area. This is illustrated in Figure 2.4b.
Typically, directional coarsening is implemented based on the relative size of the matrix coefficients with the coarsening performed every iteration. The coarsening strategy used here is a more basic approach and is done only once at the outset of the computation. As a final item, the work done to coarsen the grid increases with the use of directional coarsening. This is consistent with other observations (Mavriplis, 1995).

The AMG solver is implemented as an object-oriented design using C++. The implementation makes use of the containers and algorithms from the Standard Template Library (STL). The class diagram for AMG solver implementation is shown in Figure 2.5. The linear solution for each grid level is governed by the solver_method class. This is implemented as an abstract base class. Three linear solver classes are derived from solver_method: ILU, Gauss-Seidel (GS) and Incomplete Choleski Conjugate Gradient (ICCG). The particular solver chosen is invoked polymorphically. At present, the ILU and GS solvers are utilized with the multigrid scheme; ICCG is restricted to fine grid solutions. The multigrid solution follows a fixed V-cycle. For a typical problem the number of coarse grid levels is chosen such that the coarsest grid is comprised of ~20 cells.

Grid_parms, SquareMatrix and matrix_eqn are stand-alone classes that are used compositionally. Coarsening is handled by the grid_parms class. Experience has shown that for some flow fields the longest wavelength errors as represented on the coarsest grid must be fully eliminated. For these cases the AMG implementation allows a direct solution on the coarsest grid. The SquareMatrix class creates a square matrix representation of the coefficients on the coarsest grid to facilitate direct solution. Coefficients and source terms for all grid levels are housed within the matrix_eqn class.
2.6 Interface Sliding Technique for Rotor-Stator Computation

As mentioned before, full rotor-stator interaction is solved the all blade rows together simultaneously in code B. This is made possible by the development of a new technique that handles the information exchange from one frame of reference to another.

2.6.1 Interface Strategy

For the computation of multiple blade rows together simultaneously, multiple frames of reference are used. Relative velocity is solved in each blade row’s own frame of reference. Primary velocity variables are the absolute velocities for the stators and the relative velocities for the rotors. This approach would require the distinction of the grids between blade rows first. Then, special treatment is needed for the transfer of information from one blade row to another. This is accomplished by creating a so-called “interface” between adjacent blade rows in the solver.

There are two main considerations that should be kept in mind when generating the proper interface grids. First, it would be best that the procedure demands no interpolation. Second, it is highly desirable that the variables are solved on fixed grids, which means that the grids around the rotors should not be actually moving in the rotor frame of reference. These two considerations will determine how the interface strategy used in the code is implemented.

Recalling the wake-passing approach used in code A for time-accurate computations of single blade row cases, the grids are fixed, even for rotor passages; and the inlet profiles of velocity and turbulent quantities are moving in the circumferential direction.
The interface for multiple blade rows cases would now serve with similar purpose as the inlet boundary conditions for single blade row cases.

To illustrate the interface procedure, consider a case with two blade rows: a stator followed by a rotor downstream. Here, the blade rotation speed is presumed to be constant. The x-axis is selected to align in the direction of the rotation vector. Thus,

\[ \omega = \omega_i + 0j + 0k \quad (2.56) \]

where \( \omega > 0 \) means counter-clockwise rotation and \( \omega < 0 \) means clockwise rotation.

First, a moving grid strategy, where the grids around the rotor blades are moving, is considered. Grids are generated for the whole stage, just as if it is a case with two stators, with the exception that there exists an interface located in-between the two blade rows. For simplicity, suppose that the grids cover the full annulus of 360 degrees. From time step zero to one, the rotor blades would rotate by \( \Delta \theta = \omega \Delta t \), where \( \Delta t \) is the physical time step, and the stator blades remain still. Grids around the rotor blades are rotated by the same amount as the rotor blades, and the grids around the stator blades do not move. To separate the grids that move and those do not, an interface, which is composed of many grid faces that are continuously connected, is introduced. The interface, located entirely between the blade rows, must have a topology of a single domain, so that the grid nodes downstream of it are the grids that are rotating and the grids upstream of it are standstills.

To make matter at hand more clear, suppose the interface is actually composed of two sets of grid faces. At time step zero, each grid face belongs to the stator grids and its
vertices must have the exact locations as those belong to the rotor grids. Consider here a
set of interface faces at time step zero, \( f_1 \), which belongs to the stator grids and \( f_2 \), which
belongs to the rotor grids. Face \( f_1 \) matches \( f_2 \) exactly at time step zero. Their vertices (say,
one of the pairs be \( iv_1 \) and \( iv_2 \)) also match exactly (Figure 2.7a, in the x-\( \theta \) plane).

To illustrate it more clearly, the interface is plotted apart in Figure 2.7b, where the
dotted lines are only imaginary and are used to identify the matching faces and vertices. In
reality, the axial gap between \( f_1 \) and \( f_2 \) is zero.

At time step one shown in Figure 2.7c, \( f_2 \) is moved to a new location and there
must exist a \( f_1^* \) to match it. If it is required that there is no interpolation when transferring
information between the stator and rotor grids, \( f_1^* \) and \( iv_1^* \) must be at the exact locations
of \( f_2 \) and \( iv_2 \), respectively. The relationship between \( iv_1 \) and \( iv_2 \) (the same as \( iv_1^* \)) is:

\[
iv_1^* = iv_2 = R(iv_1)
\]  

(2.57)

where \( R \) is an operator that rotates a point by \( \Delta \theta \).

Eq. 2.57 can be repeated to reach time step \( k \) as

\[
iv_1^{(k)} = R^{(k)}(iv_1), \quad k = 1, 2, \ldots, N
\]  

(2.58)

where superscript \( (k) \) means the \( k \)-th time step, \( R^{(k)} \) means doing \( R \) operation \( k \) times,
and \( N \) is the total time steps in one unsteady period, \( T \). After \( N \) time steps,

\[
iv_1^{(N)} = iv_1^{(0)} = iv_1
\]  

(2.59)

which completes a cycle.
Eq. 2.58 requires that for any interface vertex, $iv_1$, there must exist $N - 1$ other interface vertices. These $N$ vertices must be located and evenly distributed on a circle that is perpendicular to the $x$-axis. The interface can be viewed as sets of such circle. All such sets are not required to have the same $x$ coordinate. In the case multiple circles have the same $x$ coordinate and the same radius, they can be viewed as one circle, which allows the fact that all the vertices on it could be not evenly distributed, as long as each vertex has its matching set according to Eq. 2.58.

Since all $f_1$ match their perspective $f_2$ exactly at time step zero, there would be no further requirements imposed on $f_2$. At time step $k$, $f_2$ matches exactly $f_1^{(k)}$. The information exchange happens between such pairs of faces. Recalling that in this code all the primary dependent variables are solved on the node centers, not the vertices or the faces, there is only one variable, the convection velocity $\bar{v}$, is stored on the faces. The convection velocity is the velocity flux across a face, thus it is normal to that face. On $f_1^{(k)}$, it should be the absolute convection velocity, and on $f_2$, it should be in the relative frame. In the case (valid for most axial turbomachinery situations) where the entire interface locates on a constant-$x$ plane, $\bar{v}$ contains only the axial velocity component, which is irrelevant of the frames of reference. Since $\bar{v}$ is widely used by many subroutines in the code, this feature associated with interface being on a constant-$x$ plane saves a lot of coding effects and should be preserved whenever possible.
So far, the first consideration (no interpolation for information exchange across the interface) is discussed for a strategy with moving grids. The second consideration (no moving grids) is now imposed. To accomplish this, a reverse rotation is operated on the rotor grids as illustrated in Figure 2.7d. At time step \( k \),

\[
iv_2^{(k)} = R^*(k)(iv_2) = iv_1
\]  

(2.60)

where \( R^* \) is the reverse operator of \( R \). The second equal in Eq. 2.60 means that \( f_2 \) has been moved back to its original location at time step zero and it matches \( f_1 \) exactly. So has all the rotor grids been moved back to their respectively original positions. But, the important thing here is that the transfer of information should happen between the face pair of \( f_1^* \) and \( f_2 \), not between \( f_1 \) and \( f_2 \).

In the end, no grids are actually moving. The face pairing of \( f_1^* \) and \( f_2 \) is accomplished by the shifting of the nodes the interface faces point to. In this example, \( f_{n2} \) of \( f_1 \) at time step \( k \) is actually \( k\Delta \theta \) away from its location at time step zero.

Using this approach, \( f_2 \) is just an imaginary face. There is no need for \( iv_2 \) to actually exist in the grid. It is only useful in interpreting the interface strategy here.

The essence of this strategy is that the dependent variables are solved in a way such that the rotor grids are moved by \( R^{(k)} \), then the rotor grids are rotated by \( R^*^{(k)} \) to get back to their original locations and the dependent variables on them take appropriate transformation. This sounds like a strategy involves grid moving. But in fact, it will be
shown later that with proper care, these moving back and fourth of the rotor grids are just imaginary and the strategy here is indeed one with fixed grids.

The interface faces can, of course, be any multilateral face. The currently used grid generator, GRIDGEN [25], allows only triangular and quadrilateral faces. The simplest interface is that composed of layers of quadrilateral faces, with each line in circumferential direction has constant radius and the vertices on it are distributed evenly.

This procedure requires the interface faces be set after N is set. Each time N is changed, the interface needs to be redone. To avoid overall changes in the grid, two parallel lines are drawn in each side of the interface. The upstream grid block is extended only to the parallel line upstream of the interface. The downstream grid block is extended only to the parallel line downstream of the interface. When different N is used, the grids upstream of the upstream parallel line remain the same and so do the downstream grids. The interface faces are regenerated and new domains and blocks are generated between the two parallel lines. This, in effect, requires minimum efforts.

2.6.2 Transfer of Information across Interface

Since the data structure of the solver is face based, the way a cell gets information from other cells is through faces. The faces, along with their f_n1 and f_n2 pointers that point to the two bounding cells n_1 and n_2, are what connect all the cells together. The imaginary rotating back and fourth of rotor grids simply do not have any effect on the values of scalars, and on the values of the axial component of any vector. For faces whose f_n1 and f_n2 are neither bounded by an interface face, the imaginary rotations would
have net zero effects on the y- and z-components of a vector. Thus, when looking at potential problems associated with the interface strategy and their remedies, one only needs to start from the interface faces.

At time step one, an interface face, \( f \), and its two bounding cells, \( n_1 \) and \( n_2 \), are depicted in the x-\( \theta \) plane in Figure 2.8. Compared with that of time step zero, \( n_1 \) remains at the same location and \( n_2 \) is at \( -\Delta \theta \) away. If \( n_1 \) needs information at \( n_2 \) through the interface face, or the other way around, special care must be taken when the information involves a component of a vector or tensor. For example, through face \( f \), information about an arbitrary vector \( A = (a_x, a_y, a_z) \) is taken from \( n_2 \) and given to \( n_1 \). Let’s assume it has a form:

\[
\phi_1 = c_x a_{x2} + c_y a_{y2} + c_z a_{z2}
\]  

(2.61)

where subscripts 1 and 2 represent value at \( n_1 \) and \( n_2 \), respectively, and \( c \)'s are coefficients. Values at \( n_2 \) are of course values calculated after the backward rotation. But, as far as node \( n_1 \) knows, \( n_2 \) should be directly downstream, that is, before the backward rotation. Since the code does not store values at \( n_2 \) before the backward rotation, they have to be calculated by the forward rotation, which is exactly rotating \( n_2 \) by \( R \). Denoting \( n_2^* \) as the original location of \( n_2 \) at time step zero, then

\[
a_{x2}^* = a_{x2}
\]

(2.62)

\[
a_{y2}^* = p_2 a_{y2} + q_2 a_{z2}
\]

\[
a_{z2}^* = -q_2 a_{y2} + p_2 a_{z2}
\]

where
Here, $\Delta \theta$ is not used to make Eq. 2.62 and Eq. 2.63 stand for any time step. Eq. 2.61 should then be modified as:

$$\phi_1 = c_x a_x + c_y a_y + c_z a_z$$  \hspace{1cm} (2.64)$$

On the other hand, if there is need for information be taken form $n_1$ and given to $n_2$, $n_1$ needs to be rotated backward to $n_1^{*}$ as:

$$a_{x_1}^{*} = a_{x_1}$$  \hspace{1cm} (2.65)$$

and

$$a_{y_1}^{*} = p_2 a_{y_1} - q_2 a_{z_1}$$
$$a_{z_1}^{*} = q_2 a_{y_1} + p_2 a_{z_1}$$

Since the face-node pointer for interface face changes at each time step, some geometrical quantities need modification. One of them is the distance between the face and the node it points to.

### 2.6.3 Velocity Components

For velocity components $v$ (y-direction) and $w$ (z-direction), modifications are needed not only according to Eq. 2.63 and Eq. 2.65, but also for taking into account the change from the absolute frame to the relative frame. Consider the delta-form of the momentum equation:

\[
p_2 = \frac{y_2 y_2^* + z_2 z_2^*}{r_2 r_2^*} \tag{2.63}
\]

\[
q_2 = \frac{y_2 z_2^* - z_2 y_2^*}{r_2 r_2^*}
\]
\[ A_p \Delta \phi_p = \sum A_i \Delta \phi_i + S \]  

(2.67)

where \( \phi = v, w \). For \( \Delta \phi_i \), change of reference frames has no effect since it is in delta form. For the first factor, when cell P is upstream of the interface, and cell i on the right side of Eq. 2.67, it is calculated as:

\[
\begin{align*}
\Delta v_i &= p_2 \Delta v_i + q_2 \Delta w_i \\
\Delta w_i &= -q_2 \Delta v_i + p_2 \Delta w_i
\end{align*}
\]  

(2.68)

and when cell P is downstream of the interface and cell i upstream of the interface, it becomes:

\[
\begin{align*}
\Delta v_i &= p_2 \Delta v_i - q_2 \Delta w_i \\
\Delta w_i &= q_2 \Delta v_i + p_2 \Delta w_i
\end{align*}
\]  

(2.69)

In Eq. 2.67, the source term should be calculated in a similar way as above, but the wheel speed must be taken into account. When cell P is upstream of the interface and cell i is at downstream of the interface, one should use:

\[
\begin{align*}
v_i^* &= p_2 (v_i - \Omega_x z) + q_2 (w_i + \Omega_x y) \\
w_i^* &= -q_2 (v_i - \Omega_x z) + p_2 (w_i + \Omega_x y)
\end{align*}
\]  

(2.70)

and when it is the other way around:

\[
\begin{align*}
v_i^* &= p_2 (v_i + \Omega_x z) - q_2 (w_i - \Omega_x y) \\
w_i^* &= q_2 (v_i + \Omega_x z) + p_2 (w_i - \Omega_x y)
\end{align*}
\]  

(2.71)

### 2.6.4 Gradient

As mentioned before, the calculation of scalars are not affected by the interface strategy at all. When calculating the gradient of a variable, only area of special attention is
needed for the axial gradient of \( v \) and \( w \). They should be handled in the same way as in Eq. 2.70 and Eq. 2.71.

### 2.6.5 Second Order Scheme for the Momentum Equations

Following Lien (2000), the second term in Eq. 2.22 should be treated as a vector. For \( n_1 \) is upstream of the interface and \( n_2 \) is downstream, and the source term at \( n_1 \) needs information from \( n_2 \):

\[
S_{v1} = S_{v1} - \left[ \frac{\nabla + \nabla}{2} (\nabla v \cdot \Delta r)_1 + \frac{\nabla - \nabla}{2} (\nabla v \cdot \Delta r)_2 \right] \tag{2.72}
\]

\[
S_{w1} = S_{w1} - \left[ \frac{\nabla + \nabla}{2} (\nabla w \cdot \Delta r)_1 + \frac{\nabla - \nabla}{2} (\nabla w \cdot \Delta r)_2 \right] \tag{2.73}
\]

where the last term in each of the two equations above should be modified as:

\[
(\nabla v \cdot \Delta r)_2^* = p_2[(\nabla v \cdot \Delta r)_2 - \Omega_x \Delta z] + q_2[(\nabla w \cdot \Delta r)_2 + \Omega_x \Delta y] \tag{2.74}
\]

\[
(\nabla w \cdot \Delta r)_2^* = -q_2[(\nabla v \cdot \Delta r)_2 - \Omega_x \Delta z] + p_2[(\nabla w \cdot \Delta r)_2 + \Omega_x \Delta y]
\]

Similarly, when the source term at \( n_2 \) needs information from \( n_1 \):

\[
(\nabla v \cdot \Delta r)_1^* = p_2[(\nabla v \cdot \Delta r)_1 + \Omega_x \Delta z] - q_2[(\nabla w \cdot \Delta r)_1 - \Omega_x \Delta y] \tag{2.75}
\]

\[
(\nabla w \cdot \Delta r)_1^* = q_2[(\nabla v \cdot \Delta r)_1 + \Omega_x \Delta z] + p_2[(\nabla w \cdot \Delta r)_1 - \Omega_x \Delta y]
\]

### 2.7 Frontend and Backend for Code B

Figure 2.6 shows a schematic of the frontend for the code. GRIDGEN is used to build hybrid meshes with tetrahedra, hexahedra, prisms, and pyramids. Generally geometric surfaces are covered with triangles and/or quadrilaterals. These are extruded as prisms or hexes in near wall layers, and the core flow region is filled with pyramids and tetrahedra. The extrusion process is straightforward for surfaces with nominally convex curva-
ture such as the turbomachinery blades considered below. However for geometries with highly convex curvatures or internal corners considerable handwork is required in the surface extrusion process.

GRIDGEN is also used to specify boundary conditions, where sufficient generality for our needs is provided by allowing the definition of boundary types (e.g., multiple pressure boundaries). A Fieldview [21] unstructured format file is generated by GRIDGEN.

A tool has been written (FV2METIS in Figure 2.6) to extract graph information from this file for METIS [43], which performs the domain decomposition. In particular each element in the grid is designated as a “vertex” in the graph and each element with which it shares a face represents an “edge”. kmetis is used to partition the graph into approximately equal sizes.

Additional in-house tools use the GRIDGEN and METIS outputs and split the model input files and assemble the complete geometric and boundary condition data structures for each processor.

In order to provide improved flow field initialization, a set of stringers can be specified in GRIDGEN. These are a sequence of line segments that define a nominal flow path. They are propagated through the frontend to the flow code, which initializes the velocity field by resolving a specified initial velocity magnitude into the direction of the nearest stringer segment.

A subroutine in the code outputs FIELDVIEW grid and solution files at used-specified physical time steps. Since FIELDVIEW requires solution on vertices only, a cell-to-face-to-vertex interpolation scheme is used to transfer solution on the cells to the vertices.
A separate post-processing routine handles the time-averaging and other extraction of solution at specific locations.

Solution on each boundary patch is also written to files in the code. This is especially useful for obtaining the pressure on the blade surfaces.

### 2.8 Parallelization

The code is parallelized based on domain decomposition using MPI. Partitioning is carried out in the frontend as described above. Inter-partition boundaries are input to the flow code from FVFACER as any other boundary condition with a single additional boundary patch attribute being the neighbor partition processor number. FVFACER writes inter-partition face pointers to the flow solver input file in the same order that these faces are encountered in FVSPLIT. Accordingly no reordering is required when loading and unloading 1-D structures associated with message passing.

Data is passed after each scalar is computed in the segregated procedure. For the point iterative solvers used for the scalar equations, $\Delta \phi$ is passed at every sweep of the linear solver, so that there is no degradation in convergence due to domain decomposition. For the AMG solver used for the pressure corrector equation, the code has currently not been parallelized at the matrix level. Accordingly each domain is solved independently with $p'=0$ set at inter-partition boundaries. This does give rise to a slowing of overall solver convergence especially for high L/D geometries.
Figure 2.1 Control volume with grid point notations for code A
Figure 2.2 Arbitrary polyhedral element and hierarchal data structure for code B.
Figure 2.3 Geometry nomenclature for cell face evaluations.
Figure 2.4 Coarsening of grid cells, (a) top: isotropic; (b) bottom: directional.
Figure 2.5 Class diagram for AMG solver.
Figure 2.6 Elements of the frontend for code B.
Figure 2.7 Sketches for interface sliding technique.
Figure 2.8 Sketches for interface sliding technique at time step 1.
CHAPTER 3 EFFECTS OF BLADE SPACING AND WAKE/BLADE COUNT RATIO ON THE UNSTEADY FLOW IN A COMPRESSOR STATOR

Recent trends in the design of turbomachinery are toward closer axial spacing of blade rows, increased blade loading and fewer blade rows. These result in increased flow unsteadiness and interactions between the rotor and the stator. Understanding the source of the unsteadiness and their dependency on flow and geometrical variables, and their effects on the turbomachinery performance are vital for improving the performance and design that includes these unsteady effects. The unsteady flow effects depend on the upstream wake profile, blade loading, blade row spacing, rotor/stator blade count, blade geometry of the rotor and the stator, Mach number, Reynolds number, freestream turbulence and the endwall boundary layers. The emphasis hitherto has been to understand the potential effect and viscous interaction effect. With the maturity of computational codes, parametric study can now be carried out with confidence.

One of the major concerns for designers is the aerodynamic loss due to both the blade boundary layer and the upstream wake. Major objectives of this simulation study are to investigate the effects of blade row spacing and the rotor/stator blade count on the unsteady pressure, flow field and losses, including accurate predictions of the wake transport and its properties as it progresses from upstream of the blade row to the downstream. The emphasis is not only on the flow physics, but also on the loss generation mechanism. The losses generated upstream of the blade and inside the passage are of particular interest.
3.1 Simulation Test Case

The mid-span section of the second stage stator of a two-stage compressor at the United Technologies Research Center (Stauter, et al., 1991) is chosen as the test case for the simulations. The second stage stator and its upstream rotor have 44 blades in each row. The hub/tip ratio is 0.8 with a tip radius of 0.76m. Rotation speed of the upstream rotor is 650 rpm. The compressor is operated at a flow coefficient of 0.51. The Reynolds number based on the stator blade chord length is $2.5 \times 10^5$. The corresponding reduced frequency $\omega_r$ is 8.48. Measurements of the rotor wake profile at 8% axial chord downstream of the rotor trailing edge using a Laser Doppler Velocimeter were taken in Stauter, et al. (1991). Using this profile as the inlet condition, the computation of the stator unsteady flow field and the validation of the code against these data were done in Ho and Lakshminarayana (1995). In this simulation, the same geometry is used while certain parameters are varied in order to simulate the effects of the axial spacing and the wake/blade count ratio.

The measured wake profile (Stauter, et al., 1991) at 8% axial chord downstream of the rotor (36% of axial chord upstream of the stator) for a rotor-stator spacing of 0.44C_x is shown in Figure 3.1. At this measuring plane, the maximum angle variation across the rotor wake is 22°. The predicted velocities at 13% axial chord upstream of the stator leading edge show considerable decay of the wake, with angle changes of 10° (Figure 3.2). The total velocity defect in the wake at both 44% of the axial chord and 13% of the axial chord upstream is small. The measured and predicted wake profiles are in good agreement. The major effect here is the changes in the flow angle due to a “jet” type of profile.
in the tangential gust and the “wake” type of profile for the axial velocity gust, and the resulting unsteady incidence to the stator.

The simulation includes this case ($s/C_x = 0.44$) as well as $s/C_x = 0.8$ and $0.3$. The case with $s/C_x = 0.8$ is simulated by moving the same wake, as in Figure 3.3, to 72% axial chord upstream. Likewise, $s/C_x = 0.3$ is simulated by moving the wake closer to the stator blade at $0.22C_x$. In all cases the wake is located at $0.08C_x$ downstream of the rotor.

The characteristics of the rotor wake for various simulation cases are shown in Table 3.1 for the simulations of spacing effects and Table 3.2 for the wake/blade count simulation. The simulations of the effects of the wake/blade count are carried out for the actual geometry measured by Stauter, et al. (1991). The configuration, inlet plane and the notations used are shown in Figure 3.3.

<table>
<thead>
<tr>
<th>case</th>
<th>$x_r/C_x$</th>
<th>$x_{in}/C_x$</th>
<th>$s/C_x$</th>
<th>$u_d/u_e$</th>
<th>$v_d/v_e$</th>
<th>$U_d/U_e$</th>
<th>$\alpha_d/\alpha_e$</th>
<th>r</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.08</td>
<td>0.72</td>
<td>0.80</td>
<td>0.52</td>
<td>0.24</td>
<td>0.06</td>
<td>$22^\circ$</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>0.08</td>
<td>0.36</td>
<td>0.44</td>
<td>0.52</td>
<td>0.24</td>
<td>0.06</td>
<td>$22^\circ$</td>
<td>1</td>
</tr>
<tr>
<td>3</td>
<td>0.08</td>
<td>0.22</td>
<td>0.30</td>
<td>0.52</td>
<td>0.24</td>
<td>0.06</td>
<td>$22^\circ$</td>
<td>1</td>
</tr>
<tr>
<td>4</td>
<td>0.08</td>
<td>0.36</td>
<td>0.44</td>
<td>0.00</td>
<td>0.00</td>
<td>0.06</td>
<td>$0^\circ$</td>
<td>0</td>
</tr>
</tbody>
</table>

Table 3.1 Test cases for simulations of blade row spacing effects
Table 3.2 Test cases for simulations of wake.blade count ratio

| case | \(x_r/C_x\) | \(x_{in}/C_x\) | \(s/C_x\) | \(u_d/u_c\) | \(v_d/v_c\) | \(U_d/U_c\) | \(\alpha_d/\alpha_c\) | r 
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>0.08</td>
<td>0.36</td>
<td>0.44</td>
<td>0.52</td>
<td>0.24</td>
<td>0.06</td>
<td>22°</td>
<td>1</td>
</tr>
<tr>
<td>6</td>
<td>0.08</td>
<td>0.36</td>
<td>0.44</td>
<td>0.52</td>
<td>0.22</td>
<td>0.08</td>
<td>22°</td>
<td>2</td>
</tr>
<tr>
<td>7</td>
<td>0.08</td>
<td>0.36</td>
<td>0.44</td>
<td>0.51</td>
<td>0.22</td>
<td>0.09</td>
<td>22°</td>
<td>3</td>
</tr>
</tbody>
</table>

where Case 5 is the steady state run for the second stage rotor to predict the decay of wake downstream of the rotor in the absence of the stator.

3.2 Effect of Axial Spacing between Blade Rows

As mentioned earlier, the rotor location is changed to simulate the effects of rotor/stator spacing (Table 3.1, Figure 3.3). The measured wake profile is assumed to be the same as that shown in Figure 3.1 for the case with spacing of 0.44\(C_x\). This neglects the effect of the stator pressure field on the rotor viscous flow field. The effects of the stator pressure field on the rotor wake are included from the measurement plane onwards. Thus the unsteadiness caused by the stator on the rotor wake (moving gust) is included.

In the present simulation study, the effect of varying turbulence intensity across the wake is not included. A 9% turbulence intensity is used as suggested by Stauter, et al. (1991). To capture the wake decay and the unsteady viscous flow through the stator passage accurately, 200 \(\times\) 96 grid points are used with 61 grids upstream and 72 grids inside the passage; The exception is for case 1, which has a longer inlet, the corresponding numbers are 81 and 62, respectively.
3.2.1 Unsteady Blade Pressure and Velocity Field

The time-averaged blade pressures are found to be nearly identical to the steady state pressure for all the cases. The unsteady pressures on the blade surface are Fourier decomposed to derive the first and second harmonic components, which are normalized by the tangential gust amplitude \( v_1 \),

\[
C_{pi} = \frac{p_i}{\rho v_1 U_\infty}, \quad i = 1, 2
\]  

The chordwise distribution of the first harmonic of the unsteady pressure on the suction surface is shown in Figure 3.4. The value of \( C_{p1} \) decreases, as expected, as the spacing is increased. On the pressure surface, the distribution of the first harmonic has the same trend with smaller magnitude. Hence, the pressure fluctuation on the blade, especially in the leading edge region, is higher for smaller blade row spacing. This is supported by the unsteady velocity distribution shown in Figure 3.5, which is a snap shot of the unsteady velocity corresponding to the maximum unsteady pressure fluctuations for cases 1 and 3. The vectors shown are the difference between the instantaneous velocity vector and the time-mean velocity vector.

The unsteady velocity vectors for \( s/C_x = 0.8 \) and 0.3, indicate that the velocity fluctuations are larger for the latter case. As indicated by several earlier investigators, the passage of wake results in two counter rotating vortices on either side, clockwise on the leading edge of the wake and anti-clockwise on the trailing side of the wake. Hence, before the wake impinges on the leading edge the incidence decreases, increases near the impingement location and decreases again after its passage.
The magnitude of the unsteady pressure decreases rapidly (Figure 3.4) with distance from the leading edge and this is caused by the guidance of the flow by the blade. A decrease in the streamwise velocity is predicted in the leading edge area on the suction surface with the arrival of the wake, corresponding to an increase in the local incidence and a decrease in the static pressure. Two distinct phenomena are responsible for large unsteady pressure near the leading edge, one of them is the incidence change (Figure 3.5), the other is deceleration and acceleration of the flow near the blade surface during the passage of the wake.

The unsteady pressure and flow field downstream of the leading edge is strongly affected by the recirculation or the vortex pattern. This is clear from the unsteady velocity vectors shown in Figure 3.5, as well as vorticity, where counter clockwise vorticity is positive) contours shown in Figure 3.6. The vorticity shown is defined as

\[ \Omega = \frac{\partial u}{\partial y} \frac{\partial v}{\partial x} U_\infty / C \]  

(3.2)

Large positive and negative vorticity is present on either side of the wake centerline and the vorticity associated with the wake decays rapidly downstream. The vorticity and recirculation pattern is the major source of unsteadiness in the pressure downstream of the leading edge. On the suction surface, the clockwise vortex on the leading side of the wake tends to decrease the local instantaneous velocity and increase the static pressure on the suction surface, while the counter-clockwise vortex on the trailing side of the wake tends to increase the local instantaneous velocity and decrease the static pressure. In addition, due to the orientation of the wake with respect to the blade surface, the streamwise
component of the unsteady velocity inside the wake is in the opposite direction to the
mainstream. This amplifies the effects of the clockwise vortex. The hump on the suction
side (Figure 3.4) in the 20%-30% chordwise location is caused by this effect.

\( C_{p1} \) is very small on the suction side very near the trailing edge. This is because of
the flow separation. The inception of the separation occurs approximately between 82%-90% chord, the larger spacing case has a larger flow separation region. This seems to indi-
cate that larger unsteadiness in the flow delays separation. The pressure fluctuations are
much smaller in the separated region.

It is clear from unsteady velocity vectors (Figure 3.5 and Figure 3.6) that the vorti-
ces upstream and downstream of the wake centerline are much weaker for the larger spac-
ing. Case 1, with the largest gap, has the smallest hump near 20%-30% chord (Figure 3.4).

Large variations of the phase angle at and near the leading edge for various blade
row spacing (Figure 3.4b) are caused by the differences in the convective time for these
cases. Larger rotor/stator spacing requires longer time, hence, larger phase angle differ-
ence. The phase angles in regions of very low amplitude fluctuations (e.g. trailing edge
regions) are not accurate.

The second harmonic of the pressure fluctuation on the suction surface is plotted in
Figure 3.7. The magnitude of the second harmonic for the larger gap is much smaller. This
is consistent with the decay characteristics of the wake, as the first harmonic dominates in
the far wake profile. Surprisingly, case 2, with wake farther upstream, has higher magni-
tudes than case 3 on the suction surface. This may have been caused by the non-linear
effects, shift in energy from lower to higher harmonics. The second harmonic of the phase
angle differences are similar to those of the first harmonic (Figure 3.7b). Once again, the scatter in the phase angle distribution on the pressure surface is due to low amplitudes of unsteady pressure and the associated inaccuracy.

The axial and tangential components of the total unsteady aerodynamic forces (first harmonic) for various cases investigated are shown and compared with experimental data by Gallus, et al. (1982) in Figure 3.8. It should be emphasized here that the blade geometry, configurations used in the test case here are very different from the geometry used by Gallus, et al. (1982). The comparison is shown mainly to confirm the qualitative trend and not the magnitude. The agreement with the data is reasonably good for the tangential force fluctuations. The trend and magnitude are predicted well. The agreement with the axial force fluctuations is not good. The predicted value of $F_{x1}^+$ is much smaller than the data of Gallus, et al. (1982). The measurement indicated that the force fluctuations are large in the region of $s/C_x = 0.05$ to 0.2. This region is dominated by both the potential effect (unsteady pressure interaction) and the viscous wake effect. Resolution of flow in this region has to be carried out with a multistage code, where the rotor and the stator flows are solved simultaneously.

### 3.2.2 Steady and Unsteady Blade Boundary Layer

The harmonic content of momentum thickness, $\theta$, is shown in Figure 3.9. The time-averaged values are not significantly different. Near the trailing edge, maximum values of $\theta_0$ occur for the closest blade row spacing. This is clear evidence that the profile
losses are higher for closer blade spacing. It should be cautioned here that this conclusion is valid when there is no transition.

The first harmonic of the amplitude of the momentum thickness $\theta_1$, shows that the smallest spacing has the maximum amplitude, the amplitude of fluctuations are of the order of 5% of the time-averaged value near the trailing edge. A similar trend is observed in the second harmonic, with the exception that case 2 (0.44$C_x$ spacing) has a higher second harmonic component, which is consistent with the predicted second harmonic of unsteady pressure distribution (see Figure 3.7). Furthermore, the first and second harmonics of momentum thickness are of the same order of magnitude and as expected, the second harmonic content is more sensitive to the spacing effect. This can be explained on the basis of momentum and energy transport from the first component to the second component as the wakes become more shallow and wider during the transport through the blade row.

The harmonic content of the skin friction coefficient on the suction surface is shown in Figure 3.10. The differences in $C_{f0}$ are small, except in the trailing edge region. Flow separation occurs near the trailing edge for the large blade row spacing and disappears for the smaller spacing. This is consistent with the observations made earlier. The prediction also indicates that this flow is fully turbulent. This is due to high turbulence levels at the inlet (9%). The smallest spacing has the maximum effect on the first and the second harmonic of $C_f$. It should also be observed that both the first and second harmonics are of nearly the same order of magnitude, especially for closer spacing.
3.2.3 Rotor Wake Decay and Loss Coefficient

The wake decay characteristics are plotted in Figure 3.11. The rate of decay of the wake (in the rotor reference frame) is nearly identical for all three cases of the spacing effect simulation. The rotor wake decays faster when there is stator downstream of the rotor. This is caused by the potential interaction. The mechanism responsible for the faster decay of the wake is the unsteadiness in the wake profile introduced by the stator-rotor interaction. This indicates that the unsteady interaction between the rotor and the stator increases the rate of rotor wake decay.

The aerodynamic loss coefficient is evaluated from the equation

\[ \zeta = \frac{P_{0,\text{in}} - P_0}{\frac{1}{2}\rho U_\infty^2} \]  \hspace{1cm} (3.3)

where \( P_0 \) is the local passage averaged time mean stagnation pressure and \( P_{0,\text{in}} \) is \( P_0 \) at the computational inlet plane. The passage averaged time mean total pressure is defined as:

\[ \bar{P}_0 = \frac{\int_P P_0 u_0 \, dy}{\int_S u_0 \, dy} \]  \hspace{1cm} (3.4)

where \( p \) and \( s \) stands for the pressure and suction surfaces, respectively. The distribution of loss should provide information on the losses due to wake decay upstream, inside the passage and its effect on profile losses on the blade surfaces.

The loss predictions upstream of the stator are shown for five cases in Figure 3.12a. The additional cases are stator with uniform flow (case 4), and rotor wake flow without stator (case 5). A comparison of these cases should provide information on addi-
tional losses arising from the wake decay and the loss mechanisms upstream and down-
stream of the stator leading edge. The losses upstream are mainly caused by wake decay.
The loss for case 4, is insignificant upstream of the stator, which indicates that the numerical
dissipation is very small in these computations. The interesting feature to observe is
the effect of the stator on wake losses. The presence of the stator decreases the rotor wake
defect. This is clearly evident by comparing case 2 and 5, the latter without a stator. The
losses are higher for case 2 as compared with case 5 due to faster decay of the wake and
unsteadiness caused by the stator pressure field. The trend in losses observed for cases 1, 2
and 3 are as expected. The losses are highest for case 1, which is located far upstream of
the stator, hence, losses due to wake decay are higher. The losses very near the leading
edge are not accurate due to errors in interpolating the results, hence losses are shown only
up 7 percent axial chord upstream of the stator.

The losses inside the passage are shown in Figure 3.12b. This should provide
information on the losses due to the wake decay and change in profile losses due to the
passing of the wake inside the passage. This would not include the wake losses upstream
of the stator. Case 3, with the closest blade row spacing has the highest losses. The
increase in loss is much higher than the losses due to wake decay in the absence of a stator.
Thus the non-linear and interaction effects increase losses due to upstream wakes. In addi-
tion, the smallest blade row spacing with the largest velocity defect causes the highest
losses due to the presence of strong counter-rotating vortices (Figure 3.6b). This is also
responsible for additional losses. The total loss due to wake decay alone (case without sta-
tor) is lower than the increase in loss (from steady state case) for case 2.
The total loss coefficient at the exit plane of the stator for all the cases is shown in Figure 3.13. The total losses include the loss from the computational inlet plane to the exit of the stator, and thus include losses due to the decay of the wake both upstream and inside the passage. The losses upstream of the blade leading edge are small compared with the losses inside the blade passage. As indicated earlier, the difference between case 1, 2, 3 and 4 represents additional losses due to wake decay inside the passage and the losses due to the interaction of the wake with the viscous layers inside the stator passage, this being highest for case 3 with the closest spacing. The total losses for cases 1 and 2 are smaller than the sum of total losses for cases 4 and 5. At these range of spacing between rotor and stator, the unsteadiness of the flow decreases the total loss very slightly. But when the spacing between rotor and stator goes smaller as in case 3, the total losses is found to be larger than the sum of losses for cases 4 and 5.

3.3 Effect of Wake/Blade Count Ratio

The ratio of rotor wake/stator blade count, r, is a significant parameter in the evaluation of aeromechanical and aeroacoustic performance of turbomachinery. The effect of r on unsteady performance is evaluated for r = 1, 2, 3. The wake/blade count ratio r is varied by incorporating two and three wakes within one stator passage upstream. The time-averaged properties are kept identical for all these cases by increasing the freestream velocities to keep the same mass flow and average flow inlet angles (Table 3.2). By doing so, the time mean pressure distribution on the stator blade is kept the same.
The effect of the wake/blade count is two-fold. One is the effect of the reduced frequency. It is well known that an increase in the reduced frequency $\omega_r$ decreases the unsteady pressure. The other effect is mainly caused by viscous and turbulence effects, and disturbance to the flow field caused by larger number of wakes, counter rotating vortices and the associated losses.

The first harmonic content of the unsteady pressure on the suction surface is shown in Figure 3.14. It is clear that the unsteady pressure near the leading edge decreases with an increase in the wake/blade count or an increase in the reduced frequency. The unsteady flow vectors for cases 7 and 8, shown in Figure 3.15a and Figure 3.15b, respectively, clearly reveal the interaction effects inside the passage. The wakes at higher blade count $r$ decay more rapidly due to mutual interactions; hence, the lowest blade count should have the largest influence inside the passage away from the leading edge. This is clear from Figure 3.14a. The unsteady pressure decreases inside the passage (beyond 0.25$C_x$) with an increase in the wake/blade count due to a decreased disturbance caused by the wake. As explained earlier, the interaction vortices located on the either side of the wake centerline generate the humps in the unsteady pressure distribution. These effects increase with an increase in $r$ in the region 10%-25% of chord as revealed in Figure 3.14 and Figure 3.15. This accounts for nearly similar values for $C_p$ at these locations. There are opposing mechanisms here. Higher reduced frequency reduces the unsteady pressure near the leading edge, but generates a larger number of interaction vortices, especially up to about the quarter chord point, thus increasing the unsteady pressure in this region. The appearance of the large number of humps in $C_p$ distribution at higher $r$ is due to the pres-
ence of a larger number of wakes. The phase angle distribution is nearly identical for all cases and the prediction beyond $x/C_x = 0.65$ is not accurate due to the low amplitude of fluctuations.

### 3.3.1 Steady and Unsteady Boundary Layer and Losses

The harmonic content of unsteady momentum thickness on the suction surface for various ratios of $r$, are plotted in Figure 3.16. Even though the differences are not significant, the smallest wake/blade count ratio has the highest time-averaged momentum thickness and this is consistent with the highest unsteady pressure and largest flow disturbance caused (especially beyond $x/C_x = 0.25$) at this wake/blade count ratio. This also confirms the earlier conclusion that larger unsteadiness results in higher time-averaged losses. The first harmonic, $\theta_1$, is higher for $r = 1$ beyond $x/C_x = 0.25$, this again is consistent with the distribution of the first harmonic of unsteady pressure shown in Figure 3.14a. The second harmonic, $\theta_2$, is the highest for $r = 1$, and this is attributed to the slower decay of the wake inside the passage for this case (compare Figure 3.5b and Figure 3.15b).

The chordwise distribution of aerodynamic loss is shown in Figure 3.17. The increase in aerodynamic losses with an increase in the ratio $r$ upstream of the leading edge and inside the passage is as expected. From the computational inlet plane to the leading edge, the losses increase almost exactly in proportion to the number of wakes. The losses inside the passage increase with an increase in the wake/blade count ratio. The total losses at the exit plane of the stator increase substantially with an increase in the wake/blade count. This is mainly caused by the losses due to the increased number of rotor wakes
inside the stator passage. Furthermore, the losses at all wake/blade count ratios are higher than the steady state value. Hence, the aerodynamic losses increase due to rotor-stator interaction, which strongly depends on the wake/blade count ratio.
Figure 3.1 Inlet velocity profiles for cases 1 to 4 (linear interpolation from experimental data by Stauter, et al., 1991) at 36% axial chord upstream of the stator
Figure 3.2 Rotor wake profile at 13% axial chord upstream of the stator. symbol: experimental data by Stauter, et al. (1991); line: prediction.
Figure 3.3 The second stage rotor and stator blade setting.
Figure 3.4 The first Fourier component of the suction surface unsteady pressure distribution, $C_{p1}$, (a) magnitude; (b) phase angle.
Figure 3.5 Predicted unsteady fluctuations of the velocity vector, $U(t)-U_0$. (a) case 1: $s/\ C_x = 0.80$. 

$C_x = 0.80$. 
Figure 3.5 (Cont.) Predicted unsteady fluctuations of the velocity vector, $U(t) - U_0$. (b)

case 3: $s/C_x = 0.30$. 
Figure 3.6 The predicted unsteady fluctuations of the vorticity, $\Omega(t) - \Omega_0$. (a) case 1: $s/ C_x = 0.80$. 
Figure 3.6 (Cont.) The predicted unsteady fluctuations of the vorticity, $\Omega(t) - \Omega_0$. (b)

case 3: $s/C_x = 0.30$. 
Figure 3.7 The second Fourier component of the suction surface unsteady pressure distribution, $C_{p2}$, (a) magnitude; (b) phase angle.
Figure 3.8 The first Fourier harmonic of the unsteady total blade force as a function of the spacing, \( s/C_x \), (a)axial component; (b)tangential component. symbol: experimental data by Gallus, et al. (1982); line: prediction.
Figure 3.9 The predicted momentum thickness on the suction surface, $\theta/C_x$, (a) time mean, (b) first harmonic, (c) second harmonic.
Figure 3.10 The predicted skin friction coefficient on the suction surface, $C_f$, (a) time mean, (b) first harmonic, (c) second harmonic.
Figure 3.11 Decay characteristic of the upstream rotor wake, (a) axial velocity defect; (b) maximum relative tangential velocity variation across the wake.
Figure 3.12 Distribution of the loss coefficient, (a) upstream of the stator leading edge; (b) inside the stator blade passage.
Figure 3.13 Loss coefficient at the exit of the stator as a function of spacing, s/C_x.
Figure 3.14 First Fourier component of the suction surface unsteady pressure distribution, $C_{p1}$, (a) magnitude; (b) phase angle.
Figure 3.15 Unsteady velocity vector, $U(t)-U_0$: (a) case 6: $r = 2.0$ at $t/T = 0.15$. 
Figure 3.15 (Cont.) Unsteady velocity vector $U(t) - U_0$: (b) case 7: $r=3.0$ at $t/T=0.15$. 
Figure 3.16 Momentum thickness on the suction surface, $\theta/C_x$, (a) time mean, (b) first harmonic, (c) second harmonic.
Figure 3.17 Distribution of the loss coefficient, $C_f$: (a) upstream of the stator leading edge, (b) inside the stator blade passage.
CHAPTER 4 THREE DIMENSIONAL FLOW IN A HIGH REYNOLDS NUMBER PUMP INLET GUIDE VANE

4.1 Case Description

The case investigated here is the water flow through a high Reynolds number pump (HIREP), which is measured by Zierke, et al. (1993). HIREP consists of an inlet guide vane (IGV) with thirteen blades and a rotor with seven blades. This is a constant hub-tip radius stage \( \left( \frac{r_{hub}}{r_{casing}} = 0.5 \right) \) with a tip radius of 0.5334m. The axial distance between the IGV and the rotor is about one IGV chord, which is 0.175m. The IGV has a solidity of 1.36 at the hub and 0.68 at the tip. The IGV is lightly loaded, with axial inlet flow. Based on an inlet velocity of 10.5m/s and the IGV chord, the Reynolds number is \( 2.6 \times 10^6 \). At midspan, the blade stagger angle is 7.4 degree and the outlet flow angle is about 13 degree. The variation of the blade turning angle in radial direction is very small. Measured static pressure on the IGV blade surface indicates that the blade has slightly lower loading near the endwalls than in the midspan. Detailed laser and five-hole probe data has been acquired (Zierke, et al., 1993). At 49.7% axial chord downstream of the IGV, a miniature five-hole probe is used to measure the wake profiles at eleven radial locations.

4.2 Flow Field inside the IGV Passage

4.2.1 Static Pressure Distribution on the Blade Surface

A 121x51x51 (streamwise, tangential, and radial) H-grid is generated for the IGV. The grid is clustered near the blade surface and the hub and tip walls in order to accurately
capture the boundary layers. The value of \( y^+ \) for the first grid point away from the walls is less than 2.0. Numerical experiments show that this grid size is sufficient to ensure grid-independent near wall flow resolution. The computational inlet plane is located 78% axial chord upstream of the IGV leading edge. The endwall boundary layer thickness at the inlet is set to nearly zero. The exit plane is located one chord downstream of the trailing edge of the IGV.

Figure 4.1 shows the comparison between the predicted and measured blade surface static pressure coefficients \( K_p \) defined as,

\[
K_p = \frac{p - p_{\text{ref}}}{\frac{1}{2} \rho U_{\text{tip}}^2}
\]  

(4.1)

where \( U_{\text{tip}} \) is the rotor blade speed at the tip.

The agreement between experiment and prediction is good. The distributions of the pressure coefficients at all the five radial locations are similar. On the pressure surface, the pressure decreases rapidly to about 10% chord, and is nearly constant beyond this point. On the suction surface, the pressure has a favorable gradient near the leading edge, and has a strong adverse pressure gradient near the trailing edge. This strong adverse pressure gradient causes the flow to separate on the suction surface near the trailing edge. The computation does capture the separation on the suction surface very near the trailing edge. Since the predicted pressure gradient is smaller than the measured values, the predicted separation length is also smaller. The predicted corner separation zone is also smaller than the measured one. The blade loading level is slightly lower for locations near the endwalls
than the midspan location. The incidence angle near the leading edge is found to decrease from the midspan to endwalls. This suggests three-dimensionality of the flow field as the flow approaches the blade.

4.2.2 Momentu

4.2.2 Momentum Thickness and Loss Coefficient

Figure 4.2 shows the spanwise distribution of the momentum thickness on the suction surface of the IGV at three axial locations: 50%, 75%, and 95% chord. The momentum thickness increases with axial distance, and is more pronounced near the endwalls due to the interaction between the endwall boundary layer and the blade boundary layer on the suction surface. In addition to this, the corner separation at the hub wall and the casing wall and the blade trailing edge cause the momentum thickness to increase dramatically near the blade trailing edge. It should be noted that the calculation of the momentum thickness is not accurate very near the endwalls and near the blade trailing edge. It is interesting to note that the momentum thickness near the trailing edge decreases as the wall is approached and increases dramatically very near the wall. This is due to the interaction between the corner separation, secondary flow and the blade boundary layer. In the secondary flow region, the entrainment of the fluid by the secondary vortex has beneficial effect on the blade boundary layer in the immediate vicinity of the outer edge of the boundary layer. At the first two axial locations (50% and 75% chord), the momentum thickness is small and constant radially, except in the end wall regions. But near the trailing edge, the momentum thickness increases from hub to tip away from the wall.
Figure 4.3 shows the passage-averaged aerodynamic loss coefficient distribution inside the blade passage. The loss coefficient is defined as the loss of passage- and mass-averaged total pressure coefficient,

\[ \zeta = \frac{\bar{C}_{p0} |_{le} - \bar{C}_{p0}}{\bar{\rho} \bar{v}_x \bar{\theta} ds} \]

(4.2)

where the definition of the passage- and mass-averaging for a quantity \( \phi \) is:

\[ \bar{\phi} = \frac{\int \rho v_x \phi d\theta}{\int \rho v_x d\theta} \]

(4.3)

The average loss coefficient is calculated by integrating the loss coefficient in the radial direction,

\[ \zeta_0 = \frac{\int_{\text{hub}}^{\text{tip}} \rho v_x \zeta r dr}{\int_{\text{hub}}^{\text{tip}} \rho v_x r dr} \]

(4.4)

In Figure 4.3, the \( x/C_x = 0 \) is the location of the blade leading edge. The trailing edge is at \( x/C_x = 1 \). At 4\% span, the loss is much larger than the losses at other locations shown in the figure. This is due to the interaction between the corner separation and the blade boundary layer. The losses increase almost linearly from the leading edge to the trailing edge. The jump near the trailing edge is caused by the wake mixing. The increase in loss beyond this point is small, except for the 4\% span.
4.3 Flow at the Exit of the IGV

4.3.1 Radial Distribution of the Passage-Averaged Velocities

The passage-averaged axial, tangential and radial velocity profiles at 49.7% chord axially downstream of the trailing edge of the IGV are compared with data in Figure 4.4. The predictions are in excellent agreement with the data. The boundary layer thickness at the hub and the tip walls is also predicted very well. Because of the development of the endwall boundary layers, the axial velocity near the edges of the endwall boundary layers shows slightly higher values than the axial velocity near the midspan section. Average radial velocity is very small in magnitude (Figure 4.4). Measured radial velocity shows a higher value near the tip. This is possibly due to a measurement problem mentioned by Zierke, et al. (1993). The lower radial velocity predicted near the hub is due to the lower predicted corner separation zone. The flow outlet angle ($\alpha_2$) is about 13 degrees at mid-span and is reduced several degree towards the hub and the tip (Figure 4.5). The flow turning decreases from midspan to endwalls. The agreement between the prediction and the data is good (both $v_6$ and $\alpha_2$), considering the accuracy of the measurement with a five-hole probe is $\pm 1^\circ$. The overturning very near the walls, caused by the secondary flow, is captured, even though there is no data available in this region. The underturning near the hub wall, from 5 to 20% of span, due to secondary flow and flow separation, is also captured reasonably well.
4.3.2 Wake Characteristics and Effects of Turbulence Length Scale

Figure 4.6 shows the comparison between the predicted and the measured wake profile at three typical radial locations. The radial component of the velocity inside the wake is predicted very well, including the magnitude of the inward and outward radial velocity. The tangential velocity profiles are nearly identical to those of the measurement, but with a small and nearly constant shift. This is possibly due to the assumption that the inlet flow is axial.

The largest discrepancy between the prediction and the data is in the maximum velocity defect of the axial velocity profile. The flow field is measured in all the thirteen IGV passages (Zierke, et al., 1993). Considerable variation in the wake profiles across the circumference is found. The data band, shown in Figure 4.6, shows the scatter in the maximum defect in axial velocity in the wake is very large. Although the scatter in maximum velocity defect measured is quite large, the predicted values are still larger than the peak values of velocity defect measured. This is true for all spanwise locations. Also, the predicted wake width is slightly larger than the measurement.

The source of the discrepancy may be due to both the computational and the experimental inaccuracy. The experimental data was acquired with a five hole probe of diameter 1.67mm. The spatial error due to the probe may account for some of the discrepancy. The other possibility in experimental inaccuracy is that the wakes may not be steady, they may oscillate due to the presence of the trailing vortex and the rotor downstream. Hence the probe may be averaging the wake resulting in lower axial velocity defects.
The computational inaccuracy may be due to the grid and the turbulence model used. In previous numerical investigations, it is found that freestream turbulent length scale and turbulent intensity can have appreciable influence on the wake prediction (Ho and lakshminarayana, 1996). In this calculation, a freestream turbulence intensity of 0.7 percent and a turbulence length scale of 8mm are used (according to Zierke, et al., 1993). An increase in turbulence length scale can reduce the maximum defect in the wake considerably. This is clear from the two-dimensional numerical simulation of the flow in the midspan section of the IGV carried out with three different turbulence length scales (Figure 4.7). In the present three-dimensional computation, the values of Tu and length scale suggested by Zierke, et al. (1993) are used. The predictions shown in Figure 4.7 agree with the correlation suggested by Raj and Lakshminarayana (1973). It is thus clear that the simulations are accurate. The five hole probe, with a finite size, may be averaging the velocity near the wake centerline, thus indicating lower velocity defect.

The two-dimensional profile loss coefficients predicted for these three cases are shown in Figure 4.8. As expected, with an increase in the turbulence length scale, the profile loss at the exit increases due to faster decay of the wake. The loss coefficients at the exit for two-dimensional simulations ranges from 0.025 to 0.029, which is smaller and more realistic than the three-dimensional prediction. It is possible that the radial velocity in the three-dimensional wake generate higher losses.

The IGV wake profiles are Fourier-decomposed to derive the harmonic contents. This is of interest to acousticians and designers in the prediction of the noise and vibration in the subsequent rotor blade row. The comparison between the predicted and measured
harmonic content of the wake profiles shown in Figure 4.9. Fourier coefficients with order of n which is not an integer multiple of 13 (number of IGV blades) are very small and not shown in the figure. The measured background noise floor is of the order of $10^{-3}$ $(v_n/U_{tip})$. So the Fourier harmonic with the order greater than 130 is not included in the figure either. The predicted tangential and radial components of the harmonics are in good agreement with the data. The predicted axial components of the harmonic are larger than the measured values. This is caused by the larger predicted maximum wake defect, shown in Figure 4.6.

4.3.3 Secondary Flow at the Exit

The measured and predicted secondary velocity vectors at 49.7% chord downstream of the IGV trailing edge are plotted in Figure 4.10. The plotting plane is normal to the machine axis. The secondary velocity is derived by subtracting the local circumferential-averaged tangential velocity from the velocity vector,

$$ v_s = v_r e_r + (v_\theta - \overline{v_\theta}) e_\theta $$

(4.5)

where $\overline{v_\theta}$ is the passage-averaged tangential velocity, $e_r$ and $e_\theta$ are unit vector in radial and tangential directions, respectively. The measured secondary velocities are derived from Zierke, et al. (1993). The vectors are shown are viewed from downstream to upstream.

The location of the wake center and the trailing vortex sheet are clearly seen in both the measurement and the prediction in Figure 4.10. Near the outer radius, the second-
ary vortex is clockwise, while it is counter-clockwise near the hub. In the measurement, 42% span is found to have the maximum lift. In the computation, that section is closer to 45% span. The secondary flow near the casing is fairly strong and covers nearly 30% of the span near the suction surface corner. The predictions are in reasonably good agreement with the data. The location of the secondary vortex core as well as extent is captured well. The location and magnitude of the secondary flow in the hub region is also captured well.

The distribution of deviation angle $\Delta \alpha_2 (\alpha_2 - \alpha_{2,\text{avg}})$ is plotted in Figure 4.11 for both the measurement and the prediction. Excellent agreement between the prediction and the measurement is achieved. The discrepancy between the measurement and the prediction is at most one degree. The deviation angle (due to secondary flow) is one of the most important parameters sought by designers as these influence the flow field in subsequent blade rows. The Navier-Stokes code is able to capture both the local and the average outlet angles shows its usefulness in the design and the performance analysis of pumps.

4.3.4 The Decay Characteristics of the Wake and the Secondary Flow

Both the wake and the secondary flow are major causes of unsteadiness, vibration and noise generation in the subsequent rotor blade. The level of unsteadiness depends on the magnitude of the wake depth and the distortion caused by the secondary flow. The wake and the secondary flow both decay downstream of the trailing edge. It is interesting to know which of these sources decays faster and this should provide clue to the major sources of noise and vibration in pumps. The contour of the decay of the axial velocity defect in wake ($v_{x,d}$) in the x-r plane is shown in Figure 4.12. The decay rate of the wake
in the axial direction is rapid near the trailing edge and slows down toward the exit computational plane. In the radial direction, the decay rate of the wake is almost constant over most of the span. The slowest decay rate occurs at 4-5% span from the endwalls. Similar decay pattern is observed for the tangential component of the secondary flow as shown in Figure 4.13. Here, the maximum variation of the tangential velocity in one pitch, \( v_{\theta,d} \), is chosen as the represent of the secondary flow strength. It is clear from Figure 4.12 and Figure 4.13 that the decay of the secondary flow is much slower and the noise and vibration due to this source may dominate those due to the wake.

In order to compare the decay rate of the wake \( (v_{x,d}) \) and the secondary flow \( (v_{\theta,d}) \), the velocity defects or maximum variations are normalized by their respective values at the trailing edge as shown in Figure 4.14. Three spanwise locations are shown. Near the endwalls, where the wake and secondary flow are strong, the decay of the secondary flow is slower than the decay of the wake. The mid-span wake has the fastest decay rate of all the distortions. It is thus clear that the distortions due to end wall flow persist longer than those due to the wakes away from the end walls. Hence the flows in these regions (both wakes and secondary flow) are major contributors to the unsteadiness, noise and vibration in turbomachinery.
Figure 4.1 Static pressure distribution on the blade surface of the IGV (90% span is near the tip). symbol: experimental data by Zierke, et al. (1993); line: prediction.
Figure 4.2 Momentum thickness distribution on the suction surface of the IGV.
Figure 4.3 Distribution of loss coefficient inside and downstream of the blade passage of the IGV.
Figure 4.4 Passage-averaged velocity distribution at 49.7% Cx downstream of the IGV trailing edge. symbols: experimental data by Zierke, et al. (1993); lines: predictions.
Figure 4.5 Passage-averaged yaw angle at 49.7% axial chord downstream of the IGV trailing edge. symbols: experimental data by Zierke, et al. (1993); lines: predictions.
Figure 4.6 Wake profiles at 49.7% axial chord downstream of the IGV trailing edge.
(a) 9.5% span. symbols: experimental data by Zierke, et al. (1993); lines: predictions.
Figure 4.6 (Cont.) Wake profiles at 49.7%Cx downstream of the IGV trailing edge. (b)

52.4% span. symbols: experimental data by Zierke, et al. (1993); lines: predictions.
Figure 4.6 (Cont.) Wake profiles at 49.7% Cx downstream of the IGV trailing edge. (c)

9.5% span. symbols: experimental data by Zierke, et al. (1993); lines: predictions.
Figure 4.7 Decay characteristics of the wake at mid-span due to simulations.
Figure 4.8 Loss coefficient distribution at the mid-span due to simulations.
Figure 4.9 Fourier content of the wake at 49.7% Cx downstream of the IGV trailing edge. (a) 9.5% span. symbols: experimental data by Zierke, et al. (1993); lines: predictions.
Figure 4.9 (Cont.) Fourier content of the wake at 49.7% axial chord downstream of the IGV trailing edge. (b) 52.4% span. symbols: experimental data by Zierke, et al. (1993); lines: predictions.
Figure 4.9 (Cont.) Fourier content of the wake at 49.7% axial chord downstream of the IGV trailing edge. (c) 81.0% span. symbols: experimental data by Zierke, et al. (1993); lines: predictions.
Figure 4.10 Secondary velocity vectors ($v_2$) at 49.7% chord axially downstream of the IGV trailing edge. (a) Prediction.
Figure 4.10 (Cont.) Secondary velocity vectors ($v_s$) at 49.7% chord axially downstream of the IGV trailing edge. (b) experimental data by Zierke, et al. (1993).
Figure 4.11 Flow deviation angle ($\Delta \alpha_2$ in degrees) distribution at 49.7% axial chord downstream of the IGV trailing edge. (a). Prediction.
Figure 4.11 (Cont.) Flow deviation angle ($\Delta \alpha_2$ in degrees) distribution at 49.7%Cx downstream of the IGV trailing edge. (b) experimental data by Zierke, et al. (1993)
Figure 4.12 Contour of the axial velocity defect ($v_{x,d}$) in the x-r plane.
Figure 4.13 X-r contour of the maximum variation in the tangential velocity ($v_{\theta,d}$).
Figure 4.14 Comparison of the decays of the wake and the secondary flow.
CHAPTER 5 FULLY COUPLED ROTOR-STATOR INTER-ACTION IN AN AXIAL TURBINE STAGE

The inlet wake passing strategy is used for the rotor-stator interaction case in Chapter 3 where the potential interaction between the stator and the upstream rotor is not fully modeled. The potential interaction will be addressed in this chapter for a turbine test case and the next chapter for an axial pump case with two-phase flows by solving rotors and stators together with the code B (Kunz, et al., 2001). In both cases, a stator is followed downstream by a rotor. Each flow passage through the stator and the rotor is solved in its own frame of reference. An interface is created to separate flow fields of different frames of reference. A new interface sliding technique is employed in the solver, which uses unstructured grids. It will be proven that the technique is accurate and efficient in handling such rotor-stator interactions without the need for either moving grids nor interpolations across the interface.

5.1 A Test Case: Vortex Convection

Before proceeding to the rotor-stator interaction case below, it is valuable to validate code B and assess its ability in predicting unsteady flows. The test case chosen here is the convection of the Stokes vortex. The vortex is superimposed by a uniform convection flow. The flow is assumed to be inviscid and its exact solution can be written in non-dimensionalized form as:
where \( v_0 = 0.4r \) for \( r \leq 1 \) and \( v_0 = 0.4/r \) for \( r > 1 \). Here, the vortex is rotating in the counter-clockwise direction with a core radius of unity, a core pressure of \(-0.16\) and a maximum rotating velocity of \(0.4\). With the uniform convection velocity, the vortex travels along in the positive x-direction and the flow field is unsteady.

All the results shown in this section are at the time when the vortex travels a length of 20 times the vortex core radius. Figure 5.1 shows a typical plot of the vortex in terms of the velocity vectors and the pressure field. Table 5.1 shows the comparison of the predicted vortex core pressure and the maximum rotating velocity for five cases. The computational domain has a width of 4 for cases 1 to 4 and a width of 10 for case 5. \( p_{\min} \) and \( v_{0,\max} \) are nearly identical for cases 4 and 5, thus the computational domain width of 4 is enough in alleviating the influence of the boundary conditions.

Comparing cases 1 to 4, the influence of the order of discretization schemes is clear. The core pressure is almost a quarter of the exact value of \(-0.16\) for the two cases with first order spatial scheme. For cases with second order spatial scheme, case 3 uses first order temporal scheme and loses 40% more core pressure than that of case 4, which uses second order temporal scheme.
A grid-dependence study is also carried out and the results are shown in Figure 5.2, where \( n \) is number of grid cells in one vortex core radius. Four cases with \( n = 4, 6, 10, 15 \) are computed. For the core pressure, the change is small after \( n = 6 \). The difference in \( v_{\text{max}} \) is more significant for all cases.

It is concluded from this study that second order discretizations are much more accurate than first order schemes, which should not to be used in the computations generally. Also, grid size does play a role in obtaining an accurate solution. Around 12-15 grid cells in one vortex diameter seems to be enough to achieve a grid-independent solution. This could be used as a guideline in resolving distortions such as wakes in turbomachinery computations.

### 5.2 Case Description

The Axial Flow Turbine Research Facility (AFTRF) at the Pennsylvania State University is an open circuit axial flow single stage turbine with advanced blading design. It has a constant casing radius of 0.4282m and a constant hub/casing radius ratio of

<table>
<thead>
<tr>
<th>Case</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td>order of temporal discretization</td>
<td>1</td>
<td>2</td>
<td>1</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>order of spatial discretization</td>
<td>1</td>
<td>1</td>
<td>2</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>( p_{\text{min}} )</td>
<td>-0.048</td>
<td>-0.047</td>
<td>-0.062</td>
<td>-0.100</td>
<td>-0.101</td>
</tr>
<tr>
<td>( v_{\theta, \text{max}} )</td>
<td>0.063</td>
<td>0.09</td>
<td>0.237</td>
<td>0.289</td>
<td>0.290</td>
</tr>
</tbody>
</table>

Table 5.1 Core pressure and maximum rotating velocity of the Stokes vortex.
The stage consists a nozzle with 23 blades followed downstream by a rotor with 29 blades. At the midspan, the axial chords for the nozzle and rotor are 0.1123m and 0.09294m, respectively. The axial gap between the nozzle and the rotor at the midspan is 22.6% of the nozzle axial chord. This small axial gap leads to strong unsteady interactions between the nozzle and the rotor.

At the design point, the mass flow rate is 11.03 kg/s and the rotor blade speed is 1330 rpm. Extensive experimental study was done on the AFTRF in Zaccaria (1994) and Lakshminarayana, et al. (2000). The operation condition for the experiments was set at a mass flow rate of 10.53 kg/s and a rotor blade speed at 1300 rpm. Besides measurements at various radial locations, the midspan section was examined the most thoroughly and it is investigated numerically in this chapter.

For the nozzle flow field, static pressure taps were used to obtain the steady state static pressure on the suction and pressure surfaces of the nozzle blades. The pitchwise flow field near the midchord was measured using a two-component LDV. Near the trailing edge, a miniature five-hole probe, which has a head diameter of 1.67mm, was used to measure the velocity and turbulence quantities. The same five-hole probe was used at five axial locations to measure the nozzle exit flow field including the wake profiles. At each axial location, there were 50 to 80 measuring points for one nozzle pitch.

For the rotor flow field, dynamic pressure transducers were used on the rotor blade surfaces to measure the steady and unsteady pressure in Lakshminarayana, et al. (2000). The Kulite model XCS-093 miniature sensors used have a frequency response of 40 KHz and a pressure range of 5 psia with an accuracy of 0.01 psia. Detailed measurements using
LDV were carried out at 37 axial stations spanning from upstream of the rotor blade to one chord downstream in Zaccaria (1994). There were 50 measuring points per rotor pitch in each axial station. The measurements were made at six different relative locations of the rotor with respect to the nozzle. These six locations represent six instantaneous time steps of the unsteady flow field through the rotor passage. Cycle-averaged and unresolved unsteadiness properties of velocity and turbulence quantities were obtained in Zaccaria (1994).

5.3 Grid Generation and Overall Flow Field

It is found in Zaccaria (1994) that three-dimensional effects are negligible at the midspan region. Static pressure distribution on the nozzle blade surface has little radial variations near midspan. Secondary flow and passage vortices measured downstream of the nozzle trailing edge are also small at the midspan. Downward radial velocity is present near the midspan, but this is rather caused by the imbalance between the pressure gradient and the centrifugal force and not an indication of local three-dimensionality. Therefore, it is appropriate to carry out a two-dimensional computation at the midspan section of the AFTRF stage in this chapter.

In order to reduce the computation memory and CPU time, the 23:29 nozzle to rotor blades configuration is scaled to 24:30, whose one-sixth is actually computed. So a configuration of 4 nozzle to 5 rotor blade passages is used as the computation domain. The scaling is done in such a way that the blade loadings and flow coefficients are kept the same as those of the unscaled configuration. The radius of the midspan section is kept at
0.39565m. The nozzle and rotor blades are reduced in both the axial and the circumferen-
tial directions by factors of 23/24 and 29/30, respectively. These scalings of the blade
geometry have errors of 4.2% and 3.3% for the nozzle and the rotor, respectively. The
blade count ratio error, however, is only 0.9% compared with that of the original configu-
ration. The axial gap between the nozzle and the rotor is also scaled by the same amount as
for the rotor blade. The rotation speed of the rotor blade is kept the same.

The solver used in this chapter is a three-dimensional code and uses only three-
dimensional grids. Also, the grid generator, GRIDGEN, outputs unstructured grids only in
three-dimensional form. Therefore, the grid is modeled as a thin stream sheet that centers
on the midspan section and has a thickness in radial direction of 2% of the midspan radius.
There is only one grid element in the radial direction throughout the computation domain.
A symmetry boundary condition is applied on the top and bottom surfaces of the grid in
the radial direction. Simplified radial equilibrium equations are applied to the flow exit.

5.3.1 Unstructured Grid for AFTRF Stage

An unstructured grid is generated for the whole AFTRF stage at the midspan. Fig-
ure 5.1 shows the mid-section view of the grid. The grid spans from one chord upstream of
the nozzle leading edge to one chord downstream of the rotor trailing edge. A brief
description, with emphasis on the interface region, of the grid generation process using
GRIDGEN ([25]) is given here.

First, the scaled nozzle and rotor blade profiles are loaded and O-meshes are gen-
erated around the blades on a cylindrical surface of constant radius, which equals to the
midspan radius. The surface is saved as a database entity and all the domains will be gen-
rated on this surface first. Then, a line (called connector in GRIDGEN) from the computational inlet to the exit is generated to define the lower periodic boundary. The connector is broken up at the desired interface location, which is chosen in this case to be at \( x_r/C_r = -0.081 \). Although the interface can be located much closer to the rotor blade without any problem as shown in the case studied in the next chapter, the current position is chosen to be just upstream of the first measured location for the rotor flow field at \( x_r/C_r = -0.808 \) (Zaccaria, 1994)). Two points on the lower periodic boundary line is created just a short distance upstream and downstream, respectively, of the interface point. The region that is between the two points axially and between the lower and higher periodic boundary circumferentially is defined as the interface region.

Upstream of the interface region, unstructured meshes are generated for one nozzle passage. Dense grids are used in the presumed nozzle wake region and coarser grids fill the rest of the passage (Figure 5.4). There are fifteen to twenty grid cells circumferentially in the nozzle wake region. This would ensure that there is enough spatial resolution to handle the wake decay. The rest of the nozzle exit region is relatively coarse, since the main unsteady source there is the pressure wave from the downstream rotor. Unlike the wakes, which are usually confined in a narrow region, pressure waves do not have high spatial gradient. Therefore, there is no need for fine spatial grid resolution in the non-wake portion in the nozzle exit region.

This completes one nozzle passage, which is copied three times to get the four nozzle passages with a 15° interval between adjacent passages. A similar approach is used for the creation of the five rotor passages. The difference is, however, that dense grids are
used downstream of the interface region to ensure there are enough grid elements to resolve the incoming and moving nozzle wakes.

The above generated grids, unlike the interface region, are independent of the user-specified total physical time steps in an unsteady period. The number of total physical time steps in an unsteady period is chosen to be 400, which corresponds to 2400 for the full annulus. There are 401 points that are evenly distributed on the interface connector (Figure 5.4). Unstructured grids are generated inside the interface region and the generation of surface grids on the midspan section is now complete. When it is needed that the total physical time step in a period is changed, the above generated grids can be used as the restart point. In a restart, what needs to done is just simply redimensioning the interface connector. The domains in the interface region will be automatically modified by GRIDGEN itself.

After extruding from the above surface grids in the radial direction to get three-dimensional grid elements and assigning appropriate boundary conditions, the output grid, which is in FIELDVIEW ([21]) format, will be preprocessed through FVFACER to get the final grids that can be used by the CFD solver. Since all the dependent flow variables are stored at the element centers, it would be more accurate to have the element centers have radii equal to the midspan radius. But in fact, the midspan radius is at the lower symmetry boundary for the output grid. So, in FVFACER, the radii of all the vertices are modified to let the element centers sit exactly on the midspan cylindrical plane.

The final grid has a total of 88157 elements, which consist of 15085 hexahedra and 73072 prisms. There are 2080 hexahedra and 4918 prisms for each nozzle passage and
1353 hexahedra and 10680 prisms for each rotor passage. There are about 281 prisms per rotor pitch in the interface region.

### 5.3.2 Overall Flow Field for the AFTRF Stage

The code is run on the LION-X cluster at Penn State. The machine is a Beowulf class computer and has 32 dual 500MHz nodes. The CPU time of the solver is about \(1 \times 10^{-4}\) seconds per node per psuedo-time step per processor. Ten pseudo-time steps are used in one physical time step. It takes 8 cycles, which has 8000 pseudo-time steps, to converge the solution. Figure 5.5 shows the convergence history for the last 5 physical time steps. The residuals for the axial velocity and static pressure have dropped about two orders of magnitude in ten pseudo-time steps.

Before meaningful results can be achieved, the accuracy of the interface sliding technique must be validated. Flow variables must be transported smoothly across the interface. The accuracy of the interface sliding technique is demonstrated in Figure 5.6 to Figure 5.9. The instantaneous static pressure coefficient, \(C_p\), at time step \(t/T=0.0\), is plotted for the full flow field in Figure 5.6, and near the interface region in Figure 5.7. Besides the higher values of \(C_p\) on suction surfaces of the blades and lower values on the pressure surfaces, additional pattern of the \(C_p\) associated with the interaction between the pressure wave of the rotor and the nozzle wake is found. The contour plots are smooth and the interface location can not be detected from the plot. It shows that the pressure is continuously computed across the interface.
Similar attributes can be found for the axial velocity as shown in Figure 5.8. The nozzle wake is depicted by the lower values coming out the nozzle trailing edge. The wake then meets the lower v_x region around the rotor leading edge. Throughout the region, v_x is clearly very smooth. This means that the downstream information is passed accurately to upstream.

The interface is located at about the place the nozzle wake vanishes in the figure. This could lead one to suspect that the nozzle wake is inaccurately cut off by the interface. If this happens, the depth of the nozzle wake would have been diminished suddenly, severely and incorrectly. This would in turn produce much lower unsteady response on the rotor blades and passages. But, actually this is not the case. As clearly shown in the plots of relative circumferential velocity in Figure 5.9, the wake transport is smooth across the interface. Near the interface, the wake width decrease due to the high w_\theta associated with the rotor suction surface. The minimum value of the wake increases there and then decreases afterwards. The region is more like a tide of high w_\theta, which pushes the nozzle wake up and down while the decay is smooth. And this smoothness in w_\theta answers the two question marks about the interface sliding technique, and the change in the frames of reference with different rotating speeds.

The above figures are just examples at a particular time step, the conclusions drawn above are true for any time step. However, it is difficult to show that the same conclusions hold for the time averaged flow variables. The difficulty, which is a post-processing problem, does not mean that the conclusion is wrong for the time averaged properties. Because, in theory, since the smooth transport of flow properties is valid for each and
every time step, the time averaged properties, which are drawn from all the individual time steps, must be smooth and accurate.

Figure 5.10 shows the time averaged relative flow angle, $\beta$, for the full flow domain. The interface line can be clearly seen. The reason that $\beta$ is not continuous across the interface is that the time averaging is different for the nozzle and rotor passages. As stated before, the primary unsteady frequencies are imposed by the passing of the blades and the wakes, and the velocity and pressure waves associated with them. In this computation, the blades are the four nozzle blades upstream of the interface and the five rotor blades downstream. For the rotor passages, the four nozzle wakes are the dominant unsteady force, hence the unsteady period for the rotor passages is:

$$T_r = \frac{T}{N_n} \quad (5.3)$$

where $T$ is the unsteady period corresponding to four nozzle passages or five rotor passages, and $N_n$ is the number of nozzle blades, 4. In this computation, $T_r$ represents 100 time steps since $T$ has 400 physical time steps in it. On the other hand, $T_n = T/N_r$ is the period for the nozzle passages. It represents 80 physical time steps for this particular computation.

More importantly, the interface is the physical location that separates the different frames of reference. The region upstream of it is in nozzle frame, and thus is called the nozzle passages. The region downstream of it is in rotor frame, and thus is called the rotor passages. The time averaging for each frame must be done at fixed point in the frame itself. It can be called “phase-locking” time averaging. To get a smooth time averaged
field across the interface, the entire flow field must be time averaged in either the nozzle or the rotor frame. And this is difficult for unstructured grids.

For example, if it is desired to have Figure 5.10 entirely done in the rotor frame, the averaging must be a phase-locked one for the nozzle passages. For calculating the contributions from a particular grid vertex at a particular time, the vertex must be moved circumferentially to a new location. This would always involve interpolations, with the exception that when all the grids are evenly distributed in the circumferential direction. Such interpolations are proven to be both cumbersome and inaccurate for unstructured grids, so it is not done in this case.

In the previous subsection, it is mentioned that the current interface location is chosen to have the most upstream measurement location for the rotor passages be in the rotor frame. Now the reason is clear. On one hand, the solver has the ability to handle an interface sitting very close to the rotor. In fact, such closeness is desirable in terms of computing efficiency. On the other hand, phase-locked time averaging is not accurate. Without accurate time averaged values, the unsteady fluctuations deduced can not be trusted either. So in this case, the efficiency is sacrificed a little in exchange for more accurate comparison with measurements.

It should be kept in mind that, in the following sections, the time averaging is done in each frame of reference only. All the unsteady fluctuations, taken by subtracting time averaged values from the instantaneous values, are only meaningful in that reference.

The distribution of the absolute total pressure is shown in Figure 5.11 for the entire stage. The coefficient is defined as:
Clearly, most of the decrease in $C_{p,\text{tot}}$ occurs inside the rotor passages since the fluid is exporting energy to the blade shaft. To give a sense of the overall performance of the code, the pitchwise distributions of $C_{p,\text{tot}}$ at two axial locations, at the interface and at the rotor exit, are shown in Figure 5.12. The passage averaged values derived at these two locations are -0.1285 and -4.347. At the interface, the loss is mainly due to nozzle wake. Measured $C_{p,\text{tot}}$ at exit is -4.375, which means the relative error for the prediction is 0.6%.

### 5.4 Nozzle Flow Field

#### 5.4.1 Flow inside the Nozzle Blade Passage

The time-averaged static pressure on the nozzle blade surfaces is plotted in Figure 5.13. The pressure coefficient is defined as:

$$C_p = \frac{p - p_{\text{ref}}}{\frac{1}{2} \rho U_{\text{inlet}}^2}$$  \hspace{1cm} (5.5)

where $p_{\text{ref}}$ is chosen to be the static pressure at the nozzle leading edge. In the figure, only $C_p$ on one nozzle blade is shown because the pressure distributions on all four nozzle blades have almost the identical values, as they should. The prediction agrees well with the measurement, which has little deviation from the designed pressure distributions (Zaccaria, 1994). The exception is at 20% to 50% axial chord on the suction surface, where the predicted $C_p$ is 0.5 lower than the measurements.
Also shown in Figure 5.13 is the local minimum and maximum values of unsteady \( C_p \). This is so-called predicted band, which is small except near the trailing edge on the suction surface. The unsteady fluctuation of \( C_p \) is caused by the rotor potential flow field, which is acting like a transverse gust downstream of the nozzle blade. Since the axial gap between the blade rows is small (0.226\( C_x \)), the pressure wave is felt on the nozzle blade. The time history of the unsteady fluctuations of \( C_p \) is plotted in Figure 5.14. It can be seen that most of the fluctuations happen on the suction surface near the trailing edge. A more precise measure of the unsteadiness in \( C_p \) is depicted in Figure 5.15 with the first three Fourier harmonics. The second and third order harmonics are negligible compared with the first order with the maximum \( C_{p2} \) is about 10\% of that of \( C_{p1} \). For the first harmonic, the values near the nozzle leading edge is about 10\% of the overall peak value. The flow upstream of the leading edge has very small unsteadiness and the flow angle has little derivation from the design value of zero (axial flow). This is consistent with the low band width for the first 20\% axial chord on both the suction and pressure surfaces of the nozzle in Figure 5.13.

The time averaged absolute total velocity profile near mid chord is shown in Figure 5.16. Two types of measurement techniques, LDV and five hole probe, are used. The predicted values are in good agreement with both measurements.

### 5.4.2 Nozzle Wakes

The nozzle wake profiles are measured at five axial stations downstream of the nozzle trailing edge. The last location is at 16\% axial chord downstream. Figure 5.17 pre-
sents the comparison between predicted and measured wake profiles in terms of total absolute velocity, $V/U_m$. The code generally overpredicts both the wake depth and width. The wake depth is under predicted at the first station ($x/C_x=1.007$), matches data well at $x/C_x=1.010$, then is overpredicted afterwards. The predicted depth is about twice that of the measurement at the last station. This discrepancy, however, is unlikely due to grid resolution in the wake region. The wake is located well within the fine mesh region as shown in Figure 5.18. There are about 20 grid cells in the circumferential direction covering the wake center region and this is more than enough for most CFD practice.

To further examine the wake decay characteristics, the wake profiles are Fourier-decomposed to get their harmonic contents. The profiles are fitted exactly into one nozzle pitch, which represents one unsteady period for the rotor passages. The spatial harmonics of the nozzle wake is the driving force that results in the unsteady response in the rotor flow field. Especially, there exists a strong linkage between the spatial harmonics of the nozzle wake and the temporal harmonics of the blade surface pressure of the rotor. Hence, it is important to get an understanding of the harmonics of the nozzle wakes.

The Fourier harmonic contents are sensitive to the point distribution of the profiles being analyzed. The original profiles of the computed wakes do not have an even distribution of points, and the resulting harmonics are not smooth and can be wrong sometimes. To overcome this problem, cubic spline fits are made on the original profiles and a new set of evenly distributed points and the wake velocity on them are extracted. The new profiles are then Fourier-decomposed.
Figure 5.19 shows the spatial harmonics for the nozzle wake in terms of $V/U_m$ (total absolute velocity nondimensionized by the rotor blade speed) at four axial locations downstream of the nozzle trailing edge. Harmonics of order 1 to order 20 for both the prediction and the measurement are shown. At all four locations, the first harmonic stands out and is the dominant force, with the predicted value is much larger than the measurements. The rest of the harmonics decrease gradually from $n=2$ to 20. The predicted wakes are much stronger than that of the measurement. Certainly, the unsteady response on the rotor flow field is expected to be overpredicted.

The first three harmonics of the nozzle wake are plotted against axial distance from the nozzle trailing edge in Figure 5.20. Though the prediction and the measurement have similar decay rate, it is the predicted wake depth near the trailing edge causes the discrepancy. The development of the nozzle boundary layer can have significant effect on the wake profiles. It is possible that due to the high Reynolds number two-equation model used in the computation, the core of the predicted wake near the trailing edge is much stronger than the measured values.

For the prediction, the drop from $n=1$ to $n=2$, $\Delta V = V_2 - V_1$, are 0.4, 0.35, 0.45, and 0.25 for stations 1 to 4, respectively. The respective measured values are 0.45, 0.4, 0.42, and 0.15. $\Delta V$ for both methods is generally decreasing with increasing $x$, with the exception at the third station.
5.5 Rotor Flow Field

As mentioned before, measurements were taken at 37 axial stations covering from 8\%C_r upstream of the rotor leading edge to one chord downstream of the trailing edge. The measurement data is processed with the phase-locked technique, which results in two-dimensional flow properties that are located in fixed grids in the rotor frame of reference. The computed rotor flow field discussed in this section is downstream of the interface. It also is fixed in the rotor frame of reference. This way, the two data sets have the same physical meanings and can be compared. The comparisons include steady and unsteady pressure on the blade surface, the time averaged rotor wake profiles, and the unsteady transportation of the nozzle wakes.

5.5.1 Static Pressure Distribution on the Rotor Blade Surface

Figure 5.24 shows the comparison between the measured and computed static pressure coefficient, C_p, and its unsteady envelope on the rotor blade surfaces. The envelope is defined as the local minimum and maximum C_p in a period. The measured C_p envelope is taken from Lakshminarayana, et al. (2000) and not at the same locations as the time averaged data. Unlike with compressors, the unsteady pressure response happen mostly downstream of the leading edge region. Turbine blades usually have large radius around the leading edge, thus, the potential velocity and pressure field covers a large region. This region will generally push the incoming wake aside. For compressors, the leading edge region is small, and the blade is cutting through the incoming wake. This
usually causes instantaneous incidence change, and high pressure response around the leading edge.

On the pressure surface, the maximum variation of $C_p$ is about 0.5, while it is about 2.0 on the suction surface for the predictions and 1.0 for the measured value. This overprediction of the pressure response on the suction surface is consistent with the fact that the nozzle wake depth is overpredicted.

To further look at the details of the unsteady pressure on the rotor blade surface, unsteady fluctuations $C_p' = C_p(t) - \overline{C_p}$ and its Fourier harmonics are plotted in Figure 5.25 and Figure 5.26, respectively. On the suction surface, most of the high fluctuations occur from 20% to 60% axial chord. Since the first harmonic is the dominant one compared to the second and third harmonics, the variation of $C_p$ over time at these mid chord locations has the shape close to a sin wave. It goes up and down and has minimum and maximum values. This is a direct result of the fact that the nozzle wake has a dominant first harmonic content. When $t/T_r$ goes from 0 to 1, the nozzle wake is passing in the negative theta direction. The wake first causes the local incidence to decrease, resulting in lower $C_p$, then it has larger incidence and higher $C_p$.

It is also noted that the lower $C_p$ band has a higher angle in the x-t plot than that of the higher $C_p$ band. At 20% axial chord, the phase lag between local low and high $C_p$ is about 180 degree ($\Delta t/T = 0.5$). That phase lag is decreased to about 72 degree ($\Delta t/T = 0.2$). This is an indication of the distortion of the nozzle wake along the nozzle suction surface.
As explained later, the high velocity region and the low velocity region are pushed to be closer and closer when passing downstream.

On the pressure surface, similar pattern is observed with the difference that the unsteady $C_p$ is much smaller here that those on the suction surface. But the region of significant unsteady pressure response is larger, from 20% to 90% axial chord, than that on the suction surface. The convex shape of the pressure surface tends to hug the nozzle wake segment and let it leave longer impression on the unsteady pressure response.

For the first harmonic on the suction surface, the unsteady pressure response has a local high value near the blade leading edge, then goes down and reaches a local minimum at about 10% axial chord. The measured maximum unsteady $C_p$ occurs at 30% axial chord, while the predicted high is at 45% axial chord. The magnitude of the high $C_{p1}$ is overpredicted by almost one fold. This is the result of the predicted slow decay of the nozzle wake mentioned before. On the pressure surface of the rotor, the harmonics are much smaller than those on the suction surface.

Another interesting thing to note in Figure 5.26 is that the peak locations of the harmonics go further downstream with increased order of harmonic, $n$. On the suction surface, the predicted locations of peaks for $C_{p1,2,3}$ are at 40%, 50%, and 60% axial chord, respectively. This is consistent with the observations made in Chapter 3. The decay of the wake tends to decrease the lower order harmonics and increase the higher order ones. As the wake transports downstream, the influence of the second and the third order harmonic gets stronger, thus delays the peak values to further downstream. For the measured values,
this trend is clear from first order to the second order, even with very limited number of data points.

### 5.5.2 Flow inside the Rotor Blade Passage

The predicted blade-to-blade profiles of the time averaged total relative velocity at three axial locations are compared with the measurements in Figure 5.27. The mass averaged absolute total velocity \( (V/U_m) \) and the mass averaged relative flow angle \( (\beta) \) are plotted in Figure 5.28 and Figure 5.29, respectively. The mass averaged value of a scalar, \( \phi \), is defined as:

\[
\overline{\phi} = \frac{\int \rho \phi v_x \, d\theta}{\int \rho v_x \, d\theta}
\]

The mass averaged absolute total velocity is a little lower than the measurements, especially downstream of 80% axial chord. This is due to the fact that the predicted nozzle wake depth is larger than the measured values. The relative flow angle is in good agreement with the data, except after the trailing edge where 1 to 1.5 degrees lower values are found for the prediction.

### 5.5.3 Nozzle Wakes inside Rotor Blade Passage

The nozzle wakes are the main source of unsteadiness generated on the rotor blades and inside the rotor flow passages. In the previous section, the unsteadiness on the rotor blade surface is discussed. Here, detailed description of the unsteady flow through the rotor passages will be presented. This is accomplished by viewing the flow in the rotor frame of reference. The nozzle wakes are acting like transverse gusts at a chosen location
upstream of the rotor leading edge. Particular interest is given to the transport of the nozzle wakes themselves.

To identify the traces of the nozzle wakes inside and outside the rotor passages, some general properties associated with wakes should be examined and identified as possible indicators of the nozzle wakes inside the rotor flow passages. Two of such properties are obviously the low (or high for a jet-like wake in certain situations) velocity and high turbulence intensity around the nozzle wake center. In Zaccaria (1994), the turbulence intensity, the total relative velocity, the relative flow angle and components of the Reynolds stress, are used to study the transport of the nozzle wakes. Six relative locations of the nozzle wakes to the rotor blades are presented as snapshots of the unsteady rotor flow field. These relative positions are evenly spaced circumferentially in the nozzle frame, thus represent six time steps with time interval of T/6.

Figure 5.30 shows the contours of the turbulence intensity at all six nozzle wake positions. The value shown is defined as:

\[ T_{ur} = \frac{\sqrt{k}}{W} \times 100 \quad (5.7) \]

The predicted contours are also given. It is not known in Zaccaria (1994) what are the exact relative locations of the nozzle and rotor blades, i.e., the exact relative angular coordinates, for the measuring positions. Through comparison of these plots, it is determined that the position 1 is corresponding to \( t/T = 0.50 \) for the computation. Thus, \( t=50, 67, 83, 00, 17, 33 \) are identified approximately for positions number 1 to 6, respectively.
In these plots, the experiment plot is for one rotor passage. The two passages shown are identical to each other. However, the computation plot is different. In the latter plots, it is taken from a set of five rotor passages. Thus, each passage there has a phase lag of 0.2T from its neighbor passages. The rotor blade at interest should be the second one from bottom seen in the figures. All comparisons should be done on this blade.

One important parameter to note here is the reduced frequency,

\[ \omega_r = \frac{C_r}{v_x T_r} \] (5.8)

which is the physical time measured in the number of unsteady period for the nozzle wake to travel one rotor axial chord. The actual number of \( \omega_r \) for this configuration is about 1.94, which is not affected by the blade scaling since the scaling is done by the same amount in the axial and circumferential directions. This indicates there should be close to two nozzle wakes axially inside the rotor blade passages at any time. In the following figures that are used to view the nozzle wake transport through the rotor passages, the attention is focused on a selected particular nozzle wake and one rotor blade first. The movement of the wake in one period, starting from it is just ahead of the rotor leading edge, is analyzed. The rest of the figures including the subsequent wakes are ignored first and are analyzed later.

Starting from position 2, the nozzle wake is just impinging on the rotor blade. The prediction does not show a strong \( Tu_r \) under the leading edge as the measurements indicate. The predicted wake is slanted towards the blade suction surface, while the measured one is quite straight and do not bend towards the rotor suction surface. It is found that
there is no such computed wake pattern for any the time steps. In the prediction, there always seems to be a distance away from the suction surface beyond which the nozzle wake can not penetrate.

The nozzle wake centerline is at an angle almost the same as the exit absolute flow angle for the nozzle. That angle is about 70°. In the rotor frame, the flow angle at the nozzle exit is changed to 42° (relative flow angle). But, this change from the absolute frame to the relative frame has no influence on the trajectory of the nozzle wake and the spatial orientation of the nozzle wake is still at 70° to the x-axis. Thus a 90° straight wake trajectory that covers the whole rotor pitch shown in the measurement data seems to be unrealistic. A larger view which includes the beginning of the wake from the nozzle trailing edge is not available from the measurements, which would have given more information about the wake at this time step.

Also, there is a relatively large region (about a quarter of the pitch) under the rotor suction surface that has no measured data available due to LDV measuring difficulty. For the prediction, the wake is present in that region.

At position 3, the chopping of the nozzle wake by the rotor blade is pronounced. The wake is fragmented into two main pieces. The upper portion of the wake enters the rotor passage between the pressure surface and the midpitch region. The wake trajectory here is bowed towards the midpitch region caused by the higher throughflow velocity of the midpitch region and lower velocity near the blade leading edge. The lower portion of wake is clear for the prediction but falls in the blank region for the measurement.
At position 4, the upper portion of the wake travels (faster for the prediction than the measurement) downstream and is slanted even more. In the measurement, due to the existence of the blank region the wake can not be traced back further upstream. This is not the case for the prediction. In fact, it can be seen that the next rotor blade (upper one) is just exerting influence on this wake and rendering its trajectory to be curved downwards.

At position 5, the wake is beginning to be cut off by the next blade. From here on out, the wake will be fragmented and each segment will go its separate way. This is shown more clearly at positions 6 and 1. For positions 3 to 6 back to 1, the nozzle wake trajectory is slanted less for the prediction than that of the measurements. The reason is explained here.

There are two main mechanisms responsible for the distortion of the nozzle wake inside the rotor blade passage. One is the different convection velocities in the circumferential direction. The convection velocity is higher near the blade suction surface and lower near the pressure surface. In the figures, the blade passage is located between the viewer and the rotation axis (x-axis). The distortion due to this factor is clockwise.

The other mechanism is caused by the difference in the axial velocity through the rotor passage due to blade blockage. From the leading edge to midchord, the axial velocity is increasing. This means in this region, the front end (downstream) of the nozzle wake is moving faster than the back end (upstream). The distortion of the nozzle wake is then counterclockwise, since the relative and absolute flow angels are both positive.
In this case, the two mechanisms are having opposite effects on the nozzle wake transport. The first mechanism prevails and the net effect is a clockwise distortion, as shown in both the prediction and the measurement.

The difference in the predicted and measured nozzle wake slant is possibly due to higher strength of the predicted wake. The larger depth of the predicted wake seems to be more resistant to the distortions imposed by the rotor passage flow field. This is more clearly manifested in the next figure.

Figure 5.31 presents the instantaneous distributions of the relative total velocity \(\frac{W}{U_m}\) at the six relative nozzle/rotor positions. The nozzle wake is identified as the region where the contour lines are highly skewed towards upstream. Similar patterns of the transport of the nozzle wake are observed.

At position 4, the predicted nozzle wake, starting from the nozzle trailing edge, follows the shape of the rotor suction surface for the first quarter axial chord, then enters the low speed region of the pressure surface and become less visible. For the measurement, the wake can not be seen upstream of 10% axial chord due to the blank region and the view is limited further upstream. So it is not clear where is exactly the upstream portion of the wake. Nonetheless, for the prediction, lower axial velocity at the wake center will strengthen the second distortion mechanism mentioned above. A stronger counterclockwise stretching is present and cancels out the clockwise distortion imposed by the difference in pitchwise convection velocity. Therefore, the predicted wake distortion is less.
To gain more knowledge of the nozzle wake transport in the rotor passage, the instantaneous fluctuations the relative total velocity ($W'/U_m$) is plotted in Figure 5.32. Unfortunately, similar figure is not present in Zaccaria (1994) and the original data can not be retrieved to process such a figure for comparison. So only computation results are used in Figure 5.32. Here, instead of plotting the same rotor passage at six different nozzle/rotor positions, a single plot is presented where the five successive rotor passages in the computational domain are plotted at the instantaneous time step that corresponds to position 1 for the bottom rotor blade. Here, a number, 1 to 5 from the bottom to the top, respectively, is assigned to each passage for the sake of convenience. In the rotor reference frame, the nozzle is rotating from bottom to top in the figure. Therefore, the time step for each rotor passage seen in the figure is $0.2T_r$ ahead of the passage below it.

In Figure 5.32, the nozzle wake is identified as blue (negative value). In passage 3, the wake seems to be cut off abruptly at 15% chord. But in fact, the bottom portion of the wake is continuing towards the pressure surface as can be seen in the plots for the turbulence intensity. The velocity in the wake region there is very close to the local time averaged value and therefore is not a local unsteady fluctuation. $0.4T_r$ later in passage 5, the upper portion of the wake actually has positive fluctuations as the red region below the blue region. This, however, is just caused by the effect of the change of frame of reference.

Figure 5.33 shows the absolute velocity fluctuations for the same five passages at the same time step as in Figure 5.32. Here, the wake is more visible but should not be confused with the vortex structures generated by the wake passing. At each side of the wake,
positive fluctuations, which have comparable magnitudes as local wake depth, are generated. The pattern of wake crossed by the positive fluctuation is evident staring from passage 5. It is distorted by the pitchwise velocity gradient as it is convected downstream as seen in the second half of each passage.

Figure 5.34 present the instantaneous distributions of the relative flow angle ($\beta$) at the six relative nozzle/rotor positions. The nozzle wake is most visible at positions 3 to 6 inside the rotor blade passage.

Figure 5.32 and Figure 5.33 can be also used in explaining the static pressure fluctuations presented in two subsections before. There, the pressure fluctuations are dominated by two regions, one with negative values and the other positive, in the x-t plots (Figure 5.25). For the rotor suction surface, the positive $V'$ generated on the upper-right side of the wake results in negative pressure fluctuation (blue region in Figure 5.25). The wake itself brings negative velocity fluctuation and positive pressure response. As the wake and vortex pattern travel downstream, they are pushed towards the suction surface and the distance between the high and low $V'$ peaks are closer. This distance is translated into phase lags in Figure 5.25. Closer distance downstream means less phase lag for the low and high peaks of pressure fluctuations. Therefore, the blue and red strips shown in Figure 5.25 are merging towards each other from upstream to downstream.

5.5.4 Rotor Wakes

The time averaged rotor wake profiles at five axial locations are plotted in Figure 5.35. For the wake depth, the agreement between the data and prediction is good from $x_r$/
$C_t=1.035$ on downstream. At the first two locations, the wake depth is overpredicted. It is noted in Zaccaria (1994) that the rapid decay of the measured wakes could be caused by three-dimensional effects.

For the predicted wake width, it is reasonably good on the pressure side of wake center, but the width on the suction side of the wake center is underpredicted. This is also possible due to the three-dimensional effects captured in the experiment but not modeled in the computations.

Figure 5.36 shows the time averaged flow angle at five axial stations downstream of the rotor. The flow angle change in the wake region is captured well for the first location, but seems to be smoothed out gradually for the rest of the downstream locations. This could be the work of the strong interaction between the nozzle wake and the rotor wake. As shown in the snapshots of the fluctuations of relative total velocity ($W/Um$) in Figure 5.37, the magnitude of the fluctuations is as high as $0.25Um$ in the near wake region and $0.15$ one chord downstream.
Figure 5.1 Vortex vector (maximum length in figure = 0.289) and pressure contour 
(blue = -0.10 and red = 0.01) for the Stokes vortex case 1.
Figure 5.2 Profiles of the vortex core pressure and the maximum rotating velocity for the Stokes vortex with different grid size.
Figure 5.3 Mid-section view of the unstructured grid with 4 nozzle and 5 rotor blades for the AFTRF configuration.
Figure 5.4 Mid-section view of the unstructured grid in the nozzle wake and interface regions for the AFTRF configuration.
Figure 5.5 Pseudo-time convergence history of the axial velocity and pressure for five successive physical time steps for the AFTRF rotor-stator computation
Figure 5.6 Instantaneous distribution of the static pressure coefficient ($C_p$) at $t/T=0.0$ for the AFTRF rotor-stator computation
Figure 5.7 Instantaneous distribution of the static pressure coefficient ($C_p$) near the interface region at $t/T=0.0$ for the AFTRF rotor-stator computation
Figure 5.8 Instantaneous distribution of the axial velocity ($v_x/U_m$) near the interface region at $t/T=0.0$ for the AFTRF rotor-stator computation
Figure 5.9 Instantaneous distribution of the relative tangential velocity ($w_\theta/U_m$) near
the interface region at $t/T=0.0$ for the AFTRF rotor-stator computation
Figure 5.10 Time-averaged relative flow angle ($\beta$ in degrees) for the AFTRF rotor-stator unsteady computation.
Figure 5.11 Time-averaged absolute total pressure coefficients ($C_{p,tot}$) for the AFTRF rotor-stator computation
Figure 5.12 Time-averaged absolute total pressure coefficient ($C_{p,\text{tot}}$) at the interface and rotor exit for the AFTRF rotor-stator computation
Figure 5.13 Time-averaged static pressure coefficients on the nozzle blade surfaces.

symbol: experimental data by Zaccaria (1994); line: prediction.
Figure 5.14 Unsteady fluctuations of the static pressure coefficients \( (C_p(t) - \overline{C_p}) \) on the nozzle blade surface for the AFTRF rotor-stator computation.
Figure 5.15 The first three harmonics of the static pressure coefficients on the nozzle blade surface for the AFTRF rotor-stator computation.
Figure 5.16 Time-averaged absolute total velocity distribution near the nozzle mid chord at $x/C_x=0.56$ for the AFTRF rotor-stator computation. symbols: experimental data by Zaccaria (1994).
Figure 5.17 The time-averaged profiles of the nozzle wake at five axial stations downstream of the nozzle trailing edge for the AFTRF rotor-stator computation.

symbols: experimental data by Zaccaria (1994); line: prediction.
Figure 5.18 The time-averaged profiles of the nozzle wake with grids shown for the AFTRF rotor-stator computation.
Figure 5.19 Spatial harmonic content of the nozzle wake at four axial locations downstream of the nozzle for the AFTRF rotor-stator computation. Symbols: experimental data by Zaccaria (1994); lines: prediction
Figure 5.20 Decay of the spatial harmonic content of the nozzle wake for the AFTRF rotor-stator computation. Top: the first harmonic; Bottom: the second and third harmonics; symbols: experimental data by Zaccaria (1994); lines: prediction
Figure 5.21 The time-averaged profiles of the nozzle wake absolute flow angle at three axial stations downstream of the nozzle trailing edge for the AFTRF rotor-stator computation. symbols: experimental data by Zaccaria (1994); lines: prediction
Figure 5.22 The time-averaged profiles of the nozzle wake static pressure at $x/C_x = 1.025$ for the AFTRF rotor-stator computation. Symbols: experimental data by Zac- caria (1994); lines: prediction
Figure 5.23 Predicted time-averaged static pressure distribution around the nozzle exit for the AFTRF rotor-stator computation
Figure 5.24 Time-averaged and unsteady envelope of the static pressure distribution on the rotor blade surface for the AFTRF rotor-stator computation. Symbols: experimental data by Lakshminarayana, et al. (2000); lines: prediction
Figure 5.25 Predicted unsteady fluctuations of the static pressure distribution ($C_p(t)$-$\overline{C_p}$) on the rotor blade surface for the AFTRF rotor-stator computation.
Figure 5.26 Distribution of the first three harmonics of the static pressure on the rotor blade surface for the AFTRF rotor-stator computation. symbols: experimental data by Lakshminarayana, et al. (2000); lines: prediction
Figure 5.27 Blade-to-blade distributions of the total relative velocity \((W/U_m)\) at three axial locations inside the rotor blade passage for the AFTRF rotor-stator computation. symbols: experimental data by Zaccaria (1994)
Figure 5.28 Mass-averaged total absolute velocity \((V/U_m)\) through the rotor blade passage for the AFTRF rotor-stator computation. symbols: experimental data by Zaccaria (1994); line: prediction
Figure 5.29 Mass-averaged relative flow angle ($\beta$) through the rotor blade passage for the AFTRF rotor-stator unsteady computation. symbol: experimental data by Zaccaria (1994); line: prediction
Figure 5.30 Snapshot of the turbulence intensity ($T_{u_r}$) at nozzle position 1. top: experimental data by Zaccaria (1994); bottom: prediction (contour interval is 2).
Figure 5.30 (Cont.) Snapshot of the turbulence intensity ($Tu_r$) at nozzle position 2.

top: experimental data by Zaccaria (1994); bottom: prediction (contour interval is 2).
Figure 5.30 (Cont.) Snapshot of the turbulence intensity ($T_u$) at nozzle position 3.

top: experimental data by Zaccaria (1994); bottom: prediction (contour interval is 2).
Figure 5.30 (Cont.) Snapshot of the turbulence intensity ($Tu_i$) at nozzle position 4.

top: experimental data by Zaccaria (1994); bottom: prediction (contour interval is 2).
Figure 5.30 (Cont.) Snapshot of the turbulence intensity ($T_u$) at nozzle position 5.

top: experimental data by Zaccaria (1994); bottom: prediction (contour interval is 2).
Figure 5.30 (Cont.) Snapshot of the turbulence intensity ($Tu_i$) at nozzle position 6.

top: experimental data by Zaccaria (1994); bottom: prediction (contour interval is 2).
Figure 5.31 Snapshot of the total relative velocity ($W/U_m$) at nozzle position 1. Top: experimental data by Zaccaria (1994); Bottom: unsteady prediction, where contour value interval is 0.11.
Figure 5.31 (Cont.) Snapshot of the total relative velocity ($W/U_m$) at nozzle position

2. Top: experimental data by Zaccaria (1994); Bottom: prediction, where contour
tvalue interval is 0.11
Figure 5.31 (Cont.) Snapshot of the total relative velocity ($W/U_m$) at nozzle position

3. Top: experimental data by Zaccaria (1994); Bottom: prediction, where contour value interval is 0.11
Figure 5.31 (Cont.) Snapshot of the total relative velocity ($W/U_m$) at nozzle position

4. Top: experimental data by Zaccaria (1994); Bottom: prediction, where contour value interval is 0.11
Figure 5.31 (Cont.) Snapshot of the total relative velocity ($W/U_m$) at nozzle position 5. Top: experimental data by Zaccaria (1994); Bottom: prediction, where contour value interval is 0.11
Figure 5.31 (Cont.) Snapshot of the total relative velocity ($W/U_m$) at nozzle position 6. Top: experimental data by Zaccaria (1994); Bottom: prediction, where contour value interval is 0.11
Figure 5.32 Snapshot of the predicted total relative velocity fluctuations ($W'/U_m$) at nozzle position 1
Figure 5.33 Snapshot of the predicted total absolute velocity fluctuations \((V'/U_m)\) at nozzle position 1
Figure 5.34 Snapshot of the relative flow angle at nozzle position 1. Top: experimental data by Zaccaria (1994); Bottom: prediction, where contour value interval is 7.5 deg.
Figure 5.34 (Cont.) Snapshot of the relative flow angle at nozzle position 2. Top: experimental data by Zaccaria (1994); Bottom: prediction, where contour value interval is 7.5 deg.
Figure 5.34 (Cont.) Snapshot of the relative flow angle at nozzle position 3. Top: experimental data by Zaccaria (1994); Bottom: prediction, where contour value interval is 7.5 deg.
Figure 5.34 (Cont.) Snapshot of the relative flow angle at nozzle position 4. Top: experimental data by Zaccaria (1994); Bottom: prediction, where contour value interval is 7.5 deg.
Figure 5.34 (Cont.) Snapshot of the relative flow angle at nozzle position 5. Top: experimental data by Zaccaria (1994); Bottom: prediction, where contour value interval is 7.5 deg.
Figure 5.34 (Cont.) Snapshot of the relative flow angle at nozzle position 6. Top: experimental data by Zaccaria (1994); Bottom: prediction, where contour value interval is 7.5 deg.
Figure 5.35 Time-averaged blade-to-blade distributions of the total relative velocity 
\( \frac{W}{U_m} \) for the rotor wakes at five axial locations for the AFTRF rotor-stator com-
putation. symbols: experimental data by Zaccaria (1994); lines: prediction
Figure 5.36 Time-averaged blade to blade distributions of the relative flow angle for the rotor wakes at five axial locations for the AFTRF rotor-stator computation. Symbols: experimental data by Zaccaria (1994); lines: prediction
Figure 5.37 Distributions of fluctuations of relative total velocity at five axial locations for the AFTRF rotor-stator computation.
CHAPTER 6 SINGLE-PHASE AND MULTIPHASE ROTOR-STATOR ANALYSIS OF FLOW IN AN AXIAL PUMP

6.1 Governing Equations for Multiphase Flows

The governing equations for multiphase flows are an extension to the governing equations for single phase flows (Eq. 2.1) with additions of the interface exchange terms. The continuity, momentum, k and ε equations for each phase (denoted by superscript k) respectively are:

\[
\frac{\partial}{\partial t}(\alpha_k \rho_k) + \frac{\partial}{\partial x_j}(\alpha_k \rho_k u_j^k) = \sum_{k \neq l} (\Gamma_{lk}^k - \Gamma_{kl}^k) \tag{6.1}
\]

\[
\frac{\partial}{\partial t}(\alpha_k \rho_k u_i^k) + \frac{\partial}{\partial x_j}(\alpha_k \rho_k u_j^k u_i^k) = -\alpha_k \frac{\partial p}{\partial x_i} + \alpha_k \rho_k g_i + \frac{\partial}{\partial x_j}(\alpha_k \mu_k \frac{\partial u_i^k}{\partial x_j}) - \alpha_k \rho_k \epsilon_{ijn} \omega_j \epsilon_{nlm} \omega_l x_m - 2 \alpha_k \rho_k \epsilon_{ijn} \omega_j u_n^k + M_i^{lk} + \sum_{k \neq l} [D^{kl}(u_i^k - u_i^l) + \Gamma_{lk}^k u_i^l - \Gamma_{kl}^l u_i^k] \tag{6.2}
\]

\[
\frac{\partial}{\partial t}(\alpha_k \rho_k u_i^k) + \frac{\partial}{\partial x_j}(\alpha_k \rho_k u_j^k u_i^k) = \frac{\partial}{\partial x_j}(\alpha_k \frac{u_i^k}{\rho_k} \frac{\partial}{\partial x_j}) + Pr_k \frac{\partial}{\partial x_j}(\alpha_k \frac{\partial u_k}{\partial x_j}) \tag{6.3}
\]

\[
+ \rho_k - \alpha_k \rho_k \epsilon_k + \sum_{k \neq l} [F_k D^{kl}(k^1 - k^k) + \Gamma_{lk}^l k^l - \Gamma_{kl}^k k^k]
\]

In the momentum equations, there are four classes of interface exchange terms included in the code, but only drag terms are used in the computation in this chapter.
6.2 Case Description and Grid Generation

The case studied here is the one previously described in Chapter 4, where the steady state three dimensional flow through the stator blade row of the HIREP stage is computed and compared with the measured data by Zierke, et al. (1993). Here, the rotor and stator flow fields are solved simultaneously on a midspan streamsheet, using the second code. Both single- and two-phase analyses are undertaken.

\[
\frac{\partial}{\partial t} (\alpha^k \rho^k \varepsilon^k) + \frac{\partial}{\partial x_j} \left( \alpha^k \rho^k u_j^k \varepsilon^k \right) = \frac{\partial}{\partial x_j} \left( \alpha^k \frac{\mu^k}{\rho^k} \frac{\partial \varepsilon^k}{\partial x_j} \right) + \text{terms involving pressure and stresses}
\]

(6.4)

6.3 Single-Phase Rotor-Stator Interaction

A rotor-stator analysis was performed on an assumed cylindrical midspan stream sheet. The grid, which is illustrated in Figure 6.1, employs 2288 hexahedra and 4303 prisms for each IGV passage and 3100 hexahedra and 6989 prisms for each rotor passage. The interface region employs 614 prisms per rotor pitch. Two sets of computations were performed, one with the stator scaled from 13 to 14 blades, so that a 2-stator-1-rotor simulation (2:1) could be run, the other with the full annulus (13:7).

Figure 6.2 shows the pseudo-time convergence history for five successive physical time steps for the 2:1 computation. A two-order of magnitude drop in pressure residual is achieved in 10 pseudo-timesteps.
Figure 6.3 shows four time snapshots of predicted axial velocity \( (u/U_{\text{tip}}) \) contours at \( t/T = 0.00, 0.25, 0.50, 0.75 \) (top to bottom) where \( T \) is the unsteady period \( (2\pi/14\omega) \) for this 2:1 case). Clearly observed are the well resolved IGV and rotor wakes, afforded by the unstructured grid and second order accurate convection discretization used.

In Figure 6.4 and Figure 6.5, the time averaged blade pressure coefficients,

\[
K_p = \frac{\frac{p - p_{\text{ref}}}{1/2 \rho U_{\text{tip}}^2}} \tag{6.5}
\]

are plotted for the IGV and the rotor and are compared with experimental data by Zierke, et al. (1993). Both 2:1 and full annulus predictions are shown. For the IGV, comparison between analysis and experiment is very good except near the trailing edge on the suction surface where the data shows a less steep rise in pressure than that predicted. This may be associated with an underprediction of trailing edge separation, likely due to the high Reynolds number \( k-\epsilon \) turbulence model employed. The rotor predictions are not as good, in particular there is an overprediction of loading near the suction peak. This is consistent with an overturning of the flow by the stator. For both the IGV and the rotor the time averaged blade pressures are nearly identical for the 2:1 and full annulus simulations.

Figure 6.6 shows the first five harmonics of unsteady pressure on the rotor pressure and suction surfaces. Overall, the magnitude of the harmonic decreases from the first to the fifth order. Also, the magnitudes on the suction surface are larger than those on the pressure surface. On the suction surface, there are humps whose locations move from around the mid chord for the first harmonic to about 20% chord for the fifth harmonic.
These humps are indications of the interaction of the stator wake with the rotor and their impact on the unsteady loading on the rotor blade.

6.4 Multi-Phase Rotor-Stator Interaction

The multi-phase (i.e., bubbly) flow through the midspan section of the HIREP axial flow pump stage is also investigated. This flow field is of relevance to surface ship propulsion applications. In particular, the bubbles which are generated along the ship hull-free surface boundary are transported along and underneath the hull. Much of this bubble field can pass near and/or though the propulsor. Accordingly, the bubbly wake of the ship, which is of significant interest in wake acoustic signature analysis, is significantly influenced by bubble-propulsor interactions. Though very little experimental data is available for bubbly propulsor flow, a bubbly flow analysis of the HIREP stage was pursued to demonstrate the multiphase capabilities of the second code. These results, which are presented here, represent a first demonstration of such a calculational capability that has appeared.

A multi-bubble-field analysis was performed. Specifically, separate continuity and momentum are solved for five gas "fields" each with a unique representative bubble diameter. This bubble size “binning” approach is employed to accommodate the significant variation of interfacial forces with bubble size. For example, larger bubbles encounter significantly lower drag forces than smaller bubbles which, in turn, give rise to higher liquid-bubble relative velocities arising from buoyancy, and apparent accelerations due to streamline curvature and system rotation within the carrier phase. For the relatively dis-
perse bubbly flow considered here, turbulence transport equations are solved for the liquid carrier phase only.

The inflow to ship propulsors can vary significantly in the circumferential direction due to the boundary layer which develops along the hull. If the propulsor is situated far enough below the keel of the vehicle (as is design practice for modern combatants), the velocity field encountered at the propulsor inlet plane will be close to circumferentially invariant. However, the bubble field generated at the hull is dispersed significantly as it is transported aft, giving rise to a large circumferential variation in bubble field volume fractions at the propulsor inlet plane. Accordingly, a two phase CFD analysis should accommodate this variation by specifying appropriate circumferentially varying profiles of volume fraction at the inlet to the computational domain.

In the present calculation, bubble sizes of 20, 50, 100, 200 and 500 µm, which spans the relevant characteristic sizes encountered in surface ship propulsor applications, are employed. For the six-field computation (field 1 = water, fields 2-6 = gas), the same grid was employed as in the single phase analysis presented above. Constant, purely axial inlet velocities were specified with no slip between phases. For this proof-of-capability analysis, a full-annulus was not done, rather a 2:1 reduced stage approximation was employed. A free-stream volume fraction of .001 was specified for each bubble field with a local 1/2 cosine profile increasing to a maximum of .002 with a period of π/7. This profile can be seen in Figure 6.7, which shows time snapshots of the predicted field 2 (20 µm bubbles) volume fraction at t/T=0.00, 0.25, 0.50, 0.75.
The code was run exactly as for the single stage rotor-stator simulation. After an approximately 2-cycle start-up transient the solution evolved into a time periodic flow field exhibited in Figure 6.7. Several interesting features of the two-phase flow are observed in this Figure. First, the bubble cloud is well preserved for this drag-force-only computation, a result of the second order accurate convection term discretization in code B. The bubble cloud is chopped up by the rotor which redirects it closer to axially and is then seen to interact with the rotor wake. Despite the high drag force exerted on these smaller bubbles, there is an accumulation of bubble volume fraction in the very near wake regions of both blades due to the preferential deceleration of the lower inertia gas phase. The very near wall regions (not discernible in this figure) also exhibit this increase in $\alpha^2$.

In order to demonstrate the dynamics associated with multiple bubble size simulations, Figure 6.8 illustrates contours of the difference in volume fraction between fields 4 and 2 (100 and 20 mm bubbles respectively) at $t/T=0.00, 0.25, 0.50, 0.75$. In the very near wake, this difference is positive indicating a larger value of $\alpha^4$ due to its lower drag. Significant carrier liquid acceleration in the regions just outside of the blade boundary layers gives rise to a preferential acceleration of the larger bubbles as indicated by the blue regions in Figure 6.8.
Figure 6.1 Views of hybrid grid used for HIREP rotor-stator computation.
Figure 6.2 Pseudo-time convergence history for five successive physical time steps for HIREP rotor-stator computation.
Figure 6.3 Time snapshots of predicted axial velocity ($u/U_{tip}$) at $t/T=0.00, 0.25, 0.50, 0.75$ (top to bottom) for HIREP stage.
Figure 6.4 Predicted and experimental time averaged IGV midspan blade pressure distributions for HIREP configuration. symbol: experimental data by Zierke, et al. (1993); lines: prediction.
Figure 6.5 Predicted and experimental time averaged rotor midspan blade pressure distributions for HIREP configuration. symbol: experimental data by Zierke, et al. (1993); lines: prediction.
Figure 6.6 Predicted first five harmonics of the unsteady pressure on the rotor blade surface for the HIREP rotor-stator computation.
Figure 6.7 Snapshots of predicted field $2(20\mu m)$ volume fraction at $t/T=0.0, 0.25, 0.5, 0.75$ (top to bottom) for HIREP stage with simulated hull bubble boundary layer inflow. 6-field, 2:1 stage computation. Range: red=.002, green=.001.
Figure 6.8 Snapshots of predicted $\alpha^4-\alpha^2$ (100, 20µm respectively) volume fraction at $t/T=0.0, 0.25, 0.5, 0.75$ (top to bottom) for HIREP stage with simulated hull bubble boundary layer inflow. 6-field, 2:1 stage computation. Range: red= .0003, blue= -.0003.
7.1 Concluding Remarks

Two CFD solvers are adapted and extended in this research for computations of unsteady, viscous, three-dimensional incompressible flows through turbomachinery blade rows. A wake-passing strategy and an interface sliding technique are used to resolve rotor-stator interactions. Four test cases are computed to validate the codes and to demonstrate the code’s accuracy and efficiency.

First, based on the simulation study carried out for the unsteady flow field through a second stage stator of a two-stage compressor with the inlet wake passing strategy, it is found that a decrease in the rotor-stator blade row spacing causes more decay of the upstream wake and leads to increases in the unsteady pressure fluctuations on the blade surface. The nonlinear effects and variations in the harmonic content are captured by the Navier-Stokes code. Loss mechanisms are evaluated. They include: (1) losses due to the decay of the wake upstream of the blade row; (2) losses due to the decay of the wake inside the stator passage, including modification of the wake profile by the stator pressure field; and (3) the blade profile losses, which are affected by the passing of the inlet wake. The wake/blade count ratio effect is found to be dominated by the reduced frequency effect near the stator leading edge. The wake interaction and vortex formation have major influence on the unsteady pressure response in the 10-25% axial chord region. Beyond this region, the smallest wake/blade count ratio has the most pronounced influence on the unsteady flow and pressure field. The increased interaction effect at higher wake/blade
count ratio results in a more rapid decay of the wake, thus reducing the influence of the wake beyond the 25% chord location. The losses increase significantly as the wake/blade count ratio is increased. This increase is mainly brought about by the increased number of wakes and the losses associated with their decay upstream and inside the blade passage.

Second, the three-dimensional, steady state flow field through the inlet guide vane of a high Reynolds number pump is predicted and compared with experimental data. Separations on the blade suction surface near the trailing edge and at the hub wall corner are captured, though the predicted separation zones are smaller than those measured. Loss profiles including profiles losses and wake losses are analyzed. Details of the predicted wake profiles at various radial locations are compared with measurement. Radial and tangential components of the wake profiles are captured well, but an overprediction of the wake depth is found at most radial locations. This may be due to the grid and the low Reynolds number two-equation turbulence model used in the computation. Also, it is found from a simulation study at the midspan section that the inlet turbulence length scale can have a significant effect on the decay characteristic of the wakes. Finally, analysis of the decay of the wake and the secondary flow shows that near the endwalls, the decay of the secondary flow is slower than the decay of the wake. Thus the secondary flow could be a major source of unsteadiness for the downstream blade row in these regions.

Third, the midspan section of an axial flow turbine stage is investigated. The nozzle is closely followed by a downstream rotor. The small axial gap between the blade rows indicates a strong unsteady interaction between them. Here, the rotor-stator interaction is captured by computed the nozzle and rotor flows together using the interface sliding
scheme, which allows accurate and efficient resolution of the nozzle wake and its interaction with the rotor flow field. The predicted nozzle wake decay is smaller than that measured, likely due to the two-equation turbulence model used. This leads to a higher predicted unsteady response on the rotor blade than measured. Turbulence intensity, total relative velocity and its unsteady fluctuations are used as indicators to investigate the dynamics of the nozzle wake inside and downstream of the rotor blade passage. The chopping by the rotor blade, the rapid transport of the nozzle wake along the rotor suction surface, and its interaction with the rotor wakes of the nozzle wakes are captured with reasonable accuracy.

Lastly, single- and multi-phase (i.e., bubbly) flows through the midspan section of an axial flow pump stage are investigated. This flow fields are of relevance to marine propulsion applications; in particular, the bubbly flow calculation is of interest in surface ship wake acoustic signature analysis. The hybrid unstructured, full rotor-stator, multi-bubble-field (separate continuity and momentum equations solved for five gas “fields” each with a unique representative bubble diameter) analyses performed represent a first demonstration of such a calculation capability that has appeared.

7.2 Recommendations for Future Research

Of the two codes, code B has more modern features, such as unstructured grids used, parallel computing, multi-phase flow capability, and accurate and efficient treatment of rotor-stator interactions. Therefore, it offers more potential promise for wider investiga-
tive and design applications and some recommendations for future research using code B are proposed here.

Because of the unstructured nature of the grids used, grid adaptation seems to be a logical choice for further code development. Currently, the interface sliding technique does offer an efficient handling of the rotor-stator interaction. But, grid adaptation would further enhance the solver’s efficiency by placing dense grids only around vortical disturbance inside downstream blade passages. Also, adaptive grid method has the potential for saving significant time and handwork in generating unstructured grids.

It is a general belief in the CFD community that writing the code itself is probably the easy part in establishing a reliable, accurate, and efficient analysis tool. The more time-consuming and important part is validating the code. Such attempts are made in this thesis for several test cases. But the author believes that much more work still needs to be done, especially in terms of finding cases with solid experimental data and in comparisons of models of lower fidelity (i.e., the linearized theory, Euler methods, and passage-average models). These works can be tedious and time-consuming, but they are important in terms of establishing the solver’s reliability, advantages, and application range that should be larger than those of the lower fidelity models. Validations also could shed new light on the details of the unsteady flow physics associated with vortical and potential interactions in turbomachinery flows.
REFERENCES


VITA

Wen-Sheng Yu


High School:

The Xiaogan Senior High School, Xiaogan District, Hubei Province, P.R.C.

College:

University of Science and Technology, Hefei, Anhui Province, P.R.C.

B.S. in Engineering Thermophysics.

Master’s Degree:

Institute of Engineering Thermophysics, Chinese Academy of Sciences, Beijing.

Master of Engineering in Engineering Thermophysics.

Employment Record:

Engineer, The 31st Institute, Ministry of Aeronautics and Astronautics, Beijing, P.R.C.