Inverse Design of Turbomachinery Blades
in Rotational Flow

by

Tiow Wee Teck

A thesis submitted for the degree of Doctor of Philosophy
in the Faculty of Engineering, University of London

Department of Mechanical Engineering
University College London
June 2000
To my parents
ACKNOWLEDGEMENTS

My sincere appreciation goes to Dr. Mehrdad Zangeneh for his invaluable advice and guidance throughout my post-graduate work. I am most grateful to him for taking me under his supervision and giving me the opportunity to learn from him. Special thanks also to Ebara Research Company Limited (Japan) for providing both financial and technical support for the current work, and in particular, to Dr. Akira Goto, Dr. Hideomi Harada and Mr. Hiroyoshi Watanabe for sharing their expertise in designing turbomachines and putting the codes developed to practical use in the company.

I am also deeply indebted to University College London for granting me the UCL Singapore Scholarship, and to the Society of Underwater Technology and Lee Foundation (Singapore) in providing scholar awards for my undergraduate studies. The staff and lecturers in the department of Mechanical Engineering have been most helpful during my stay at UCL. Professor Roy Collins and Dr. Christopher Nightingale deserve my special thanks. They encouraged me to do a research degree following my undergraduate studies and were always generous in giving advice and encouragement. My sincere gratitude is expressed to them.

Additionally, I would like to thank Dr. Jianwei Zhou for giving sound advice in writing the thesis and for tuition in the mathematical aspects of fluid dynamics. The sharing of the in-depth knowledge of optimization techniques by Dr. Cedric Yiu is also much appreciated. It has been a pleasure collaborating with him to extend the current methods with an optimization algorithm. Thanks are also due to Mr. Indi Tristanto for helping to perform the validation of the 3-D flow solver code with the commercial code, TASCFlow.

Finally, thanks to my family for enduring much hardship in order to provide me with a good education and for always encouraging me to pursue my interests despite their personal sacrifices. All the test cases are dedicated to you.

Tiow Wee Teck
June 2000
ABSTRACT

This thesis describes the development of two inverse design methods, suitable for designing two- and fully three-dimensional turbomachinery blades. Integral to both methods is a finite volume, time-marching solver of the unsteady Euler equations. The solver was developed with an accurate shock capturing technique and improved with viscous modelling. Extensive verification of the flow solution was carried out on both subsonic and transonic test cases, which include the NASA-designed transonic fan, rotor 67. Good correlation was obtained between the computation and the experiment.

The first design method developed is based on the mass-averaged swirl velocity distribution and computes the required blade shape directly from the discrepancies between the target and initial distributions. The method designs blades/blade sections with finite thickness and the only assumptions are those of the flow solver, which is used in its original form without modification. The method was validated and applied to the design of both two- and three-dimensional turbomachinery cascades in transonic flow, where it was demonstrated that the design parameter can be used to control the development of high speed flow and shock formation.

The second method is based on the surface static pressure loading distribution. The basic concept of this method is to modify the original blade wall condition in the flow solver and impose the target specification explicitly on the surfaces of the blade or blade section, which are modelled as permeable. The permeable wall allows transpiring flow normal to the surfaces, which is used to update the blade shape. This method is also not restricted to blades with zero thickness and was validated and applied to the design of both two- and three-dimensional turbomachinery cascades in high transonic flow. The NASA Lewis rotor 67 was redesigned using this method, concentrating on eliminating undesirable flow characteristics, one of which was the strong shock formation at the tip of the blade. An overall qualitative improvement was observed in the new blade.
# CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>ACKNOWLEDGEMENTS</td>
<td>3</td>
</tr>
<tr>
<td>ABSTRACT</td>
<td>4</td>
</tr>
<tr>
<td>LIST OF FIGURES</td>
<td>11</td>
</tr>
<tr>
<td>LIST OF TABLES</td>
<td>20</td>
</tr>
<tr>
<td>NOMENCLATURE</td>
<td>21</td>
</tr>
<tr>
<td>CHAPTER 1 INTRODUCTION</td>
<td>26</td>
</tr>
<tr>
<td>1.1 OVERVIEW</td>
<td>26</td>
</tr>
<tr>
<td>1.2 THE INVERSE DESIGN METHODOLOGY</td>
<td>28</td>
</tr>
<tr>
<td>1.3 FLOW ANALYSIS</td>
<td>30</td>
</tr>
<tr>
<td>1.4 SCOPE AND OBJECTIVES OF CURRENT STUDY</td>
<td>32</td>
</tr>
<tr>
<td>CHAPTER 2 LITERATURE SURVEY</td>
<td>34</td>
</tr>
<tr>
<td>2.1 &quot;CUT-AND-TRY&quot; DESIGN METHOD</td>
<td>34</td>
</tr>
<tr>
<td>2.2 OPTIMIZATION METHODS</td>
<td>36</td>
</tr>
<tr>
<td>2.3 INVERSE METHODS</td>
<td>39</td>
</tr>
<tr>
<td>2.3.1 Methods Based on Potential Flow Equations</td>
<td>39</td>
</tr>
<tr>
<td>2.3.1.1 Conformal Mapping Methods</td>
<td>40</td>
</tr>
<tr>
<td>2.3.1.2 Potential and Stream Function Methods</td>
<td>41</td>
</tr>
<tr>
<td>2.3.1.3 Hodograph Methods</td>
<td>43</td>
</tr>
<tr>
<td>2.3.1.4 Fictitious Gas Methods</td>
<td>44</td>
</tr>
<tr>
<td>2.3.1.5 Taylor Series Methods</td>
<td>45</td>
</tr>
</tbody>
</table>
2.3.1.6 Singularity Methods
2.3.1.7 Another Method Based on the Specification of $r\vec{V}_g$

2.3.2 Methods Based on Euler Equations
  2.3.2.1 Movable and Impermeable Surface Model
  2.3.2.2 Surface Pressure Transpiration Model
  2.3.2.3 Euler-Based Circulation Method
  2.3.2.4 Methods Based on $\Delta P$

2.3.3 Methods Based on Navier-Stokes Equations

2.4 SUMMARY OF REVIEWS

CHAPTER 3 DEVELOPMENT OF THE TIME-MARCHING EULER SOLVER

3.1 INTRODUCTION
3.2 THE GOVERNING EQUATIONS AND UNDERLYING ASSUMPTIONS
3.3 FINITE-VOLUME METHOD: CELL-VERTEX APPROACH
3.4 ARTIFICIAL VISCOSITY
3.5 NUMERICAL SCHEME: MULTI-STAGE RUNGE-KUTTA METHOD
3.6 GRID-SEQUENCING
3.7 VISCous MODELLING
3.8 GRIDDING AND BOUNDARY CONDITIONS
3.9 CONVERGENCE CRITERIA

CHAPTER 4 VALIDATION OF THE EULER FLOW SOLVER

4.1 INTRODUCTION
4.2 UTRC STATOR AND ROTOR CASCADE
  4.2.1 Description of the Experiment and Experimental Data
CHAPTER 5 THEORY OF THE DESIGN METHODS

5.1 THEORY OF METHOD I: SPECIFICATION OF $r\vec{v}_\theta$

5.1.1 Description of the Procedure
5.1.2 Background Theory
5.1.3 Derivation of the Camber Line Update Algorithm:
   Two Dimensions
   5.1.3.1 Initial Condition: Fixed Stacking Point
5.1.4 Derivation of the Camber Line Update Algorithm:
   Three Dimensions
   5.1.4.1 Initial Condition: Fixed Stacking Line
   5.1.4.2 Coordinate Transformation
5.1.5 Three-Segment Method of Defining the Target Distributions
5.1.6 Predicting an Initial Blade Camber Geometry
5.1.7 Design Convergence Criteria
5.2 THEORY OF METHOD II: SPECIFICATION OF $\Delta P$

5.2.1 Description of the Procedure

5.2.2 Transpiration Model

5.2.2.1 Modification of the Euler Solver:
   Two Dimensions

5.2.2.2 Modification of the Euler Solver:
   Three Dimensions

5.2.3 Derivation of the Camber Line Update Algorithm:
   Two Dimensions

5.2.4 Derivation of the Camber Line Update Algorithm:
   Three Dimensions

5.2.4.1 Coordinate Transformation

5.2.5 Defining the Target $\Delta P$ Distribution

5.2.5.1 Secant Loop: Design for Specific Work

5.2.6 Predicting an Initial Blade Camber Geometry

5.2.7 Design Convergence Criteria

CHAPTER 6 APPLICATIONS OF DESIGN METHOD I

6.1 APPLICATIONS TO TWO-DIMENSIONAL CASCADES

6.1.1 Design Validation: UTRC Rotor

6.1.2 Generic Compressor Rotor Cascade

   6.1.2.1 Details of Computation

6.2 APPLICATIONS TO THREE-DIMENSIONAL BLADES

6.2.1 Design Validation: MEL Turbine Nozzle

6.2.2 Redesign of the MEL Turbine Nozzle

6.2.3 Redesign of the NASA Rotor 67

   6.2.3.1 Near the Hub
   6.2.3.2 Near the Tip
   6.2.3.3 At the Mid-Span
   6.2.3.4 Overview of the Aerodynamic Changes
6.2.3.5 Discussion: Design Difficulty near the Endwalls 189
6.2.3.6 Discussion: $r\tilde{V}_\theta$ for Surface Flow Design 192

6.3 CONCLUDING REMARKS 194
6.3.1 Two-Dimensional Tests 194
6.3.2 Three-Dimensional Tests 195

CHAPTER 7 APPLICATIONS OF DESIGN METHOD II 197

7.1 APPLICATIONS TO TWO-DIMENSIONAL CASCADES 198
7.1.1 Design Validation: UTRC Stator 198
7.1.2 Generic Turbine Stator Cascade 200
7.1.2.1 Computation Using the Secant Loop 206

7.2 APPLICATIONS TO THREE-DIMENSIONAL BLADES 210
7.2.1 Design Validation: NASA Rotor 67 210
7.2.2 Redesign of NASA Rotor 67 212
7.2.2.1 Design FD3D-R1 214
7.2.2.2 Design FD3D-R2 217
7.2.2.3 Overview of the Aerodynamics Changes 220
7.2.2.3.1 High Flow Incidence at the Hub 220
7.2.2.3.2 Mach Number and Shock Formation 222
7.2.2.4 Performance Maps: Original and Redesigned Fans 227

7.3 CONCLUDING REMARKS 231
7.3.1 Two-Dimensional Tests 231
7.3.2 Three-Dimensional Tests 231

CHAPTER 8 CONCLUSIONS AND SUGGESTIONS FOR
FURTHER WORK 233

8.1 THE FLOW SOLVER 233
8.2 METHOD I: BASED ON THE SPECIFICATION OF $r\tilde{V}_\theta$ 234
8.3 METHOD II: BASED ON THE SPECIFICATION OF $\Delta P$ 236
8.4 CONTRIBUTION TO THE INVERSE DESIGN
OF TURBOMACHINERY BLADES 237

8.5 SUGGESTIONS FOR FURTHER WORK 239

8.5.1 Enhancement of Convergence Acceleration and
Improvement of the Grid System 239

8.5.2 Application to Radial Turbomachinery Blades with Splitters 240

8.5.3 Version for Incompressible Flow 241

8.5.4 Application to Multi-stage Design 241

8.5.5 Inverse Optimization Method 242

APPENDICES

APPENDIX I: EFFECT OF GRID SEQUENCING:
SAMPLE TEST CASE (UTRC STATOR) 243

APPENDIX II: DIFFUSION FACTOR 246

APPENDIX III: GRID DEPENDENCY TEST FOR THE FLOW
COMPUTATION OF NASA LEWIS ROTOR 67 249

APPENDIX IV: TRANSFORMATION METRICS AND JACOBIAN 253

APPENDIX V: DISCRETIZING THE BLADE UPDATE EQUATION:
METHOD I 256

APPENDIX VI: DISCRETIZING THE BLADE UPDATE EQUATION:
METHOD II 258

APPENDIX VII: THREE-SEGMENT METHOD 262

APPENDIX VIII: BLADE THICKNESS DISTRIBUTIONS 266

REFERENCES 268
# LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure 2.1: Illustration of Profile Shape Change by Moving the Bezier Points (Reproduced from Havakechian and Greim, 1999)</th>
<th>Page Number</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>35</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Figure 2.2: Geometry Model: Basic Parameters and Bezier Curves (Reproduced from Pierret, 1997)</th>
<th>Page Number</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>36</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Figure 2.3: Conformal Mapping</th>
<th>Page Number</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>40</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Figure 2.4: Profile with Open Trailing-Edge or Fish-Tail Shape</th>
<th>Page Number</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>52</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Figure 2.5: Summary of Existing Inverse Methods</th>
<th>Page Number</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>59</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Figure 3.1: Cell-Vertex Scheme</th>
<th>Page Number</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>64</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Figure 3.2: Turbomachinery Cascade Represented in H-Grid Topology</th>
<th>Page Number</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>72</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Figure 4.1: United Technologies Research Center Large Scale Rotating Rig (Schematic Diagram Reproduced from Dring et al., 1981 Report)</th>
<th>Page Number</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>77</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Figure 4.2: Computational Meshes for UTRC Stator and Rotor Cascades</th>
<th>Page Number</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>79</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Figure 4.3: Validation: UTRC Stator Comparison of Numerical and Experimental Pressure Coefficient, $C_p$</th>
<th>Page Number</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>80</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Figure 4.4: Validation: UTRC Rotor Comparison of Numerical and Experimental Pressure Coefficient, $C_p$</th>
<th>Page Number</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>80</td>
</tr>
</tbody>
</table>
Figure 4.5: Working Section of the Annular Cascade
(Schematic Diagram Reproduced from AGARD Report 275) 82

Figure 4.6: Meridional View of the Blade Assembly
(Schematic Diagram Reproduced from AGARD Report 275) 84

Figure 4.7: Computational Mesh for the Annular Nozzle 85

Figure 4.8: Surface Static Pressure Distribution near Hub Pressure
Tapping Location 87

Figure 4.9: Surface Static Pressure Distribution near Mid-Span Pressure
Tapping Location 87

Figure 4.10: Surface Static Pressure Distribution near Casing Pressure
Tapping Location 87

Figure 4.11: Computed Mach Number Contours at 4%, 50% & 96%
Blade Height (Contour Interval: 0.05) 88

Figure 4.12: Surface Velocity at the Hub 90

Figure 4.13: Meridional View of Rotor 67 Showing Laser Anemometer
and Aerodynamic Survey Locations
(Schematic Diagram Reproduced from AGARD Report 275) 92

Figure 4.14: Three-Dimensional Grid of NASA Rotor 67 94

Figure 4.15: Rotor 67 Design Speed Operating Characteristics 96

Figure 4.16: Total Pressure Profiles at Aero Station 1 & 2 97

Figure 4.17: Total Temperature Profiles at Aero Station 2 97

Figure 4.18: Swirl Angle Profiles at Aero Station 2 97

Figure 4.19: Stream-wise Distributions of Relative Mach Number
at the Near-Peak Efficiency Condition 100-101
Figure 4.20: Experimental (left) and Computed (right) Contours of Relative Mach Number at the Near-Peak Efficiency Condition; Contours at Intervals of 0.05; Dashed Lines are Estimated Shock Positions from Pierzga and Wood (1985) 103

Figure 4.21: Experimental (left) and Computed (right) Contours of Relative Mach Number at the Near-Stall condition; Contours at Intervals of 0.05; Dashed Lines are Estimated Shock Positions from Pierzga and Wood (1985) 104

Figure 4.22: Comparisons of Mach Number Contours: Inviscid, Inviscid/Viscous and Viscous Solutions 106

Figure 4.23: Comparisons of Computed Surface Static Pressure: Inviscid, Inviscid/Viscous and Viscous Solutions 107

Figure 5.1: Simplified Flow Chart of Design Method I 111

Figure 5.2: Blade Cascade with Prescribed Tangential Thickness 114

Figure 5.3: Stacking Condition: A Fixed Point along the Camber Line 119

Figure 5.4: Physical Meridional Mesh (Showing also the Stacking Line) Mapped into the Rectangular "Computational" Mesh 122

Figure 5.5: Three-Segment Specification 126

Figure 5.6: Fore- and Aft-Loaded Distributions 126

Figure 5.7: Simplified Flow Chart of Design Method II 133

Figure 5.8: Secant Algorithm: Finding ΔP for a Required Specific Work 142
Figure 6.1: Validation of 2-D Method I: Reproducing UTRC Rotor Profile 147

Figure 6.2: Validation of 2-D Method I: Target and Obtained Mass-Averaged Tangential Velocity Distributions 148

Figure 6.3: Surface Mach Number Distributions of a Supercritical (Controlled Diffusion) Blade (Reproduced from Hobbs and Weingold, 1984) 149

Figure 6.4: Characteristics of the Initial Cascade 152

Figure 6.5: Mass-Averaged Loading Distributions: Initial, Intermediate and Final 153

Figure 6.6: Relative Mach Number Contours: Initial, Intermediate and Final (Contour Interval: 0.015) 154

Figure 6.7: Relative Surface Mach Number Distributions: Initial, Intermediate and Final 154

Figure 6.8: Comparison of Profile Geometries: Initial, Intermediate and Final 155

Figure 6.9: Convergence Histories: Design 158

Figure 6.10: Convergence Histories: Overall Analysis 158

Figure 6.11: Comparison of Profile Geometries: Strict and Less Stringent Criteria on Flow Analysis 160

Figure 6.12: Validation of 3-D Method I: Target Mass-Averaged Swirl Velocity Distributions 161

Figure 6.13: Validation of 3-D Method I: Reproducing MEL Annular Nozzle 163

Figure 6.14: Original Mass-Averaged Loading Distribution 165

Figure 6.15: Mass-Averaged Swirl Velocity Distributions at the Hub: Original and Redesign 166
Figure 6.16: Mass-Averaged Loading Distributions at the Hub: Original and Redesign

Figure 6.17: Modification of the Mass-Averaged Swirl Velocity Distributions at the Tip: Original and Redesign

Figure 6.18: Specification for the Redesign: Mass-Averaged Swirl Velocity and Corresponding Loading Distributions

Figure 6.19: Comparison of Blade Profiles at the Hub: Original and Redesign

Figure 6.20: Surface Velocity at the Hub: Original and Redesign

Figure 6.21: Contours of Entropy Generation at the Hub: Original and Redesign

Figure 6.22: Original and Redesigned Nozzle Blades

Figure 6.23: Mass-Averaged Loading Distributions of NASA Rotor 67: Strong Shock near the Tip Region

Figure 6.24: Redesign Specification: Mass-Averaged Loading Distribution (Fore-Loaded Characteristic)

Figure 6.25: Original and Redesigned Blades

Figure 6.26: Relative Velocity Vectors at the Hub: Original Blade

Figure 6.27: Mass-Averaged Loading Distributions at 20% Blade Height: Original and Redesign

Figure 6.28: Comparison of Blade Profiles at the Hub: Original and Redesign

Figure 6.29: Relative Velocity Vectors at the Hub: Redesigned Blade
Figure 6.30: Relative Mach Number Contours and Velocity Vectors at the Tip: Original Blade

Figure 6.31: Mass-Averaged Loading Distributions and Relative Mach Number Contours at 80% Blade Height: Original and Redesigned

Figure 6.32: Comparison of Blade Profiles at 80% Blade Height: Original and Redesigned

Figure 6.33: Relative Mach Number Contours and Velocity Vectors at the Tip: Redesigned Blade

Figure 6.34: Mass-Averaged Loading Distributions and Relative Mach Number Contours at 50% Blade Height: Original and Redesigned

Figure 6.35: Comparison of Blade Profiles at 50% Blade Height: Original and Redesigned

Figure 6.36: Meridional Views of the Surface Relative Mach Number Contours (Contour Interval: 0.025)

Figure 6.37: Surface Static Pressure and Corresponding Surface Static Pressure Difference (\(\Delta P\)) Distributions: Original and Redesigned

Figure 6.38: Radial Profiles of Mass-Averaged Swirl Velocity at the Trailing-Edge: Original, Specified and Final

Figure 6.39: Modifying the Geometry Near the End-Casing

Figure 6.40: Comparison of Pressure Difference (\(\Delta P\)) Distributions: Actual and Theory
Figure 7.1: Validation of 2-D Method II: Reproducing the UTRC Stator Profile 198

Figure 7.2: Validation of 2-D Method II: Target and Final Surface Static Pressure Difference Distributions 199

Figure 7.3: Design Results: Mach Number Contours, Surface Static Pressure and Pressure Loading Distributions 203-205

Figure 7.4: Details of Computation: Variation of the Net Change between the Inlet and Outlet Mass-Averaged Tangential Velocities 207

Figure 7.5: Details of Computation: Variation of the Peak Pressure Loading 207

Figure 7.6: Initial, Intermediate and Final Geometries 208

Figure 7.7: Initial, Intermediate and Final Pressure Loading Distributions 208

Figure 7.8: Validation of 3-D Method II: Target Surface Static Pressure Loading Distribution 210

Figure 7.9: Validation of 3-D Method II: Reproducing the NASA Rotor 67 211

Figure 7.10: Original $\Delta P$ Distribution (With Relative Mach Number Contours at Three Blade Heights) 213

Figure 7.11: Redesign Specification: Design FD2D-R1 (Aft-Loaded) 214

Figure 7.12: Design FD3D-R1: Original and Redesigned Blade Geometries, Surface Pressure and Pressure Loading Distributions 216

Figure 7.13: Redesign Specification: Design FD2D-R2 (Fore/Middle-Loaded) 217

Figure 7.14: Design FD3D-R2: Original and Redesigned Blade Geometries, Surface Pressure and Pressure Loading Distributions 219

Figure 7.15: Pressure Loading Distributions at the Hub: Original and Redesigns 220

17
Figure 7.16: Relative Velocity Vectors at the Hub:
Original Fan, Designs FD3D-R1 and FD3D-R2 221

Figure 7.17: Relative Mach Number Contours at the Tip:
Original Fan, Designs FD3D-R1 and FD3D-R2
(Showing Relationships to the Static Pressure Distributions) 223

Figure 7.18: Relative Velocity Vectors near Trailing-Edge at the Tip:
Original Fan, Designs FD3D-R1 and FD3D-R2 224

Figure 7.19: Contours of Entropy Generation, exp(−ΔS/R) at the Tip:
Original Fan, Designs FD3D-R1 and FD3D-R2
(Contour Interval: 0.01) 225

Figure 7.20: Contours of Entropy Generation, exp(−ΔS/R) at the Mid-Pitch:
Original Fan, Designs FD3D-R1 and FD3D-R2
(Contour Interval: 0.01) 226

Figure 7.21: Adiabatic Efficiency:
Original Fan, Designs FD3D-R1 and FD3D-R2 227
(* Commercial Navier-Stokes Solver Code)

Figure 7.22: Total Pressure Ratio:
Original Fan, Designs FD3D-R1 and FD3D-R2 228

Figure 7.23: Relative Velocity Vectors at the Hub:
Original Fan, Designs FD3D-R1 and FD3D-R2
(Approximately at the Stalled Mass Flow of the Original Fan) 230

Figure 8.1: A Centrifugal Impeller with Splitter Blades 240

Figure 8.2: Mesh Representation of a Two-Stage Turbine
(Reproduced from Denton, 1990) 241
Appendices:

Figure I.1: Grid Change on UTRC Stator Cascade 244
Figure I.2: Convergence Histories on Analysis 245

Figure II.1: Variation of Friction Loss Versus Diffusion Factor
(Reproduced from Lieblein, 1965) 247

Figure III.1: Comparisons of Radial Profiles of Total Pressure Ratio
at Aero Station 2 251

Figure VII.1: 3-Segment Loading Distribution 263
Figure VII.2: Corresponding Swirl Velocity Distribution 264

Figure VIII.1: DCA Thickness Profile 266
Figure VIII.2: C4 Thickness Profile 267
## LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>Table 4.1:</td>
<td>Basic Specifications of the UTRC Stator and Rotor</td>
<td>76</td>
</tr>
<tr>
<td>Table 4.2:</td>
<td>Essential Features of the Annular Nozzle Blade</td>
<td>81</td>
</tr>
<tr>
<td>Table 4.3:</td>
<td>Basic Specifications of the NASA Rotor 67</td>
<td>91</td>
</tr>
<tr>
<td>Table 6.1:</td>
<td>Details of the Design Process</td>
<td>156</td>
</tr>
<tr>
<td>Table 6.2:</td>
<td>Details of the Two Design Runs: Strict and Less Stringent Criteria on Flow Analysis</td>
<td>159</td>
</tr>
<tr>
<td>Table 7.1:</td>
<td>Design Specifications: Three-Segment and Parabolic Distributions</td>
<td>201</td>
</tr>
<tr>
<td>Table 7.2:</td>
<td>Angle of Incidence at Design Point</td>
<td>229</td>
</tr>
<tr>
<td>Appendices:</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Table I.1:</td>
<td>Summary of Computation</td>
<td>244</td>
</tr>
<tr>
<td>Table III.1:</td>
<td>Grid Dependency Test: Single Parameter Check</td>
<td>250</td>
</tr>
<tr>
<td>Symbol</td>
<td>Definition</td>
<td></td>
</tr>
<tr>
<td>------</td>
<td>------------</td>
<td></td>
</tr>
<tr>
<td>$a$</td>
<td>speed of sound</td>
<td></td>
</tr>
<tr>
<td>$A$</td>
<td>area of each face of a computational cell</td>
<td></td>
</tr>
<tr>
<td>$A_r, A_\theta, A_z$</td>
<td>projected areas of the faces of a cell in the $r, \theta, z$ directions</td>
<td></td>
</tr>
<tr>
<td>$AP$</td>
<td>axial position of the measurement plane of the laser anemometry experiment for the NASA rotor 67</td>
<td></td>
</tr>
<tr>
<td>$c$</td>
<td>blade chord</td>
<td></td>
</tr>
<tr>
<td>$c_p$</td>
<td>specific heat capacity at constant pressure</td>
<td></td>
</tr>
<tr>
<td>$c_v$</td>
<td>specific heat capacity at constant volume</td>
<td></td>
</tr>
<tr>
<td>$C_f$</td>
<td>surface skin friction coefficient</td>
<td></td>
</tr>
<tr>
<td>$C_p$</td>
<td>pressure coefficient</td>
<td></td>
</tr>
<tr>
<td>$CFL$</td>
<td>Courant, Friedrichs, Levy number</td>
<td></td>
</tr>
<tr>
<td>$d$</td>
<td>artificial viscosity terms at each face of a computational cell</td>
<td></td>
</tr>
<tr>
<td>$D$</td>
<td>artificial viscosity terms of a computational cell</td>
<td></td>
</tr>
<tr>
<td>$D_F$</td>
<td>diffusion factor</td>
<td></td>
</tr>
<tr>
<td>$e$</td>
<td>total energy</td>
<td></td>
</tr>
<tr>
<td>$\exp()$</td>
<td>exponent of the term within the brackets</td>
<td></td>
</tr>
<tr>
<td>$E, F, G$</td>
<td>terms giving inviscid fluxes</td>
<td></td>
</tr>
<tr>
<td>$f$</td>
<td>blade camber (tangential coordinate in 2-D or angular coordinate in 3-D)</td>
<td></td>
</tr>
<tr>
<td>$f^\pm$</td>
<td>upper, + and lower, - surfaces of the blade as defined by $f$ and $t_\theta$</td>
<td></td>
</tr>
<tr>
<td>$f_r, f_\theta, f_z$</td>
<td>viscous terms in $r, \theta, z$ directions</td>
<td></td>
</tr>
<tr>
<td>$h_\theta$</td>
<td>total enthalpy</td>
<td></td>
</tr>
<tr>
<td>$H$</td>
<td>conserved variables</td>
<td></td>
</tr>
<tr>
<td>$J$</td>
<td>Jacobian</td>
<td></td>
</tr>
<tr>
<td>$k^{(2)}, k^{(4)}$</td>
<td>constants for second and fourth order artificial viscosity terms</td>
<td></td>
</tr>
<tr>
<td>$L.E.$</td>
<td>blade leading-edge</td>
<td></td>
</tr>
<tr>
<td>$m$</td>
<td>meridional direction</td>
<td></td>
</tr>
</tbody>
</table>
\[ m_{\text{ratio}} \] ratio of the mass flow rate with respect to the choked value

\[ \text{max}(\cdot) \] taking the maximum of the terms contained in the brackets

\[ M \] Mach number

\[ \hat{n} \] unit normal vector

\[ N \] number of blades in cascade arrangement

\[ P \] static pressure

\[ P_0 \] total pressure

\[ Q \] flux change

\[ rV_{\theta} \] swirl velocity (radius multiplied by tangential velocity)

\[ r\tilde{V}_{\theta} \] mass-averaged swirl velocity

(3-D design parameter for Method I)

\[ R \] gas constant

\[ Re \] Reynolds number

\[ RP \] radial position of the measurement plane in the laser anemometry experiment for the NASA rotor 67

\[ s \] blade pitch

\[ S_i \] inviscid source terms

\[ S_{vis} \] viscous source terms

\[ t_{\theta} \] tangential blade thickness (metre in 2-D and radian in 3-D)

\[ T.E. \] blade trailing-edge

\[ U \] tangential velocity to simulate blade rotation in 2-D (blade speed)

\[ US, LS \] upper and lower surfaces of a cascade

\[ V \] absolute velocity

\[ Vol \] volume of a computational cell

\[ V_m \] meridional velocity

\[ V_{N} \] velocity normal to the face of a computational cell

\[ V_x, V_y \] velocity components in \( x, y \) directions

\[ \tilde{V}_y \] mass-averaged tangential velocity

(2-D design parameter for Method I)

\[ V_r, V_\theta, V_z \] velocity components in \( r, \theta, z \) directions
$V_{\xi}, V_{\eta}$ velocity components in $\xi$, $\eta$ directions

$W^\theta$ friction velocity

$W$ relative velocity

$W^*$ dimensionless velocity, $W/\theta W$

$\dot{W}_T$ specific work

$(x, y)$ Cartesian coordinates

$Y$ distance from the wall

$Y^*$ dimensionless wall distance, $W^\theta Y/\nu$

$(r, \theta, z)$ cylindrical polar coordinates

$|\vec{A}|$ spectral radii

$\alpha^+$ angular offset of a point on the upper, + and lower, - surfaces of a blade

$\Delta$ denotes difference in the flow value between two reference points as indicated by the relevant symbol and subscript (if no subscript is specified, e.g. $\Delta P$, the blade difference value is implied)

Note: $\Delta t$ and $\Delta S$ have different definitions.

$\Delta P$ pressure difference between the upper, + and lower, - surfaces of a blade at a given axial position (design parameter for Method II)

$\Delta S$ entropy change

$\Delta t$ time step

$\Delta f$ geometrical modification of the camber line in a design iteration

$(\xi, \eta)$ the body-fitted coordinate system (mesh quasi-streamlines defined by $\xi =$ constant, mesh quasi-orthogonals by $\eta =$ constant)

$\gamma$ ratio of specific heat capacities

$\rho$ density

$\tau_w$ wall shear stress

$\nu$ shock sensor term

$\nu$ kinematic viscosity

$\omega$ rotational speed

$\Psi$ specific head rise

23
\[ \frac{d\tilde{V}_y}{dx} \] mass-averaged loading (2-D)

\[ \frac{\partial r\tilde{V}_\theta}{\partial m} \] mass-averaged loading (3-D)

**Subscripts**

approx. approximate value

AERO2 value measured at aero station 2 in the NASA rotor 67 experiment

bl average flow values on the upper and lower surfaces of a blade at a given position

choke value at the choking condition

exit\_inlet difference between values at the exit and inlet

i,j,k or ijk grid indices of the nodes at the corners of a computational cell

\( i + \frac{1}{2}\) denotes the position between the nodes at \( i \) and \( i+1 \)

(\( j \) and \( k \) also apply)

inlet value at the inlet of cascade

islip value calculated based on satisfying the slip (tangency) flow condition

\( j_{\text{stack}} \) stacking position: fixed camber line geometry at a chosen position

(\( j_{\text{stack}} \) is the initial condition to solve for new blade camber geometry)

LE value at the blade leading-edge

max maximum value

\( n \) index of the cells surrounding a computational node

\( (n = 1 \ to \ 4 \ for \ 2-D \ and \ n = 1 \ to \ 8 \ for \ 3-D) \)

outlet value at the outlet of cascade

stall value at stall

STA1 value measured at station 1 in UTRC experiment

STA2 value measured at station 2 in UTRC experiment

TE value at the blade trailing-edge

TE\_LE difference between values at the blade trailing- and leading-edges

\( \xi \) derivative with respective to \( \xi \) (for \( r, z, f \) and \( t_\theta \) only)

\( \eta \) derivative with respective to \( \eta \) (for \( r, z, f \) and \( t_\theta \) only)
Superscripts

$m$  original value of surface pressures before the specified $\Delta P$ is imposed  
(used for method II, section 5.2, Chapter 5)

$m+1$ new value of surface pressures after the specified $\Delta P$ is imposed  
(used for method II, section 5.2, Chapter 5)

$n$  $n^{th}$ design iteration

$N$  update in the secant method  
(used for method II, section 5.2.5.1, Chapter 5)

$0$  initial value

$1$  first approximation

$(1)$ to $(4)$ stages in the 4-stage Runge-Kutta scheme

$+$  value on the upper surface of a blade

$-$  value on the lower surface of a blade

$*$  target value (used to denote the specified design value)

Overbars

$-$  circumferential average

$\sim$  mass-average

Abbreviations

AGARD  Advisory Group for Aerospace Research and Development
ANN  artificial neural networks
CFD  computational fluid dynamics
GA  genetic algorithm
GM  gradient based method
LA  laser anemometer
MEL  Marchwood Engineering Laboratories
NASA  National Aeronautics and Space Administration, USA
N-S  Navier-Stokes equations
SA  simulated annealing
UTRC  United Technologies Research Center
CHAPTER 1
INTRODUCTION

1.1 OVERVIEW

The complex flow behaviour through a turbomachine presents a major challenge in the
design of efficient turbo-components. The main working component in a turbomachine
is its blades, which change the total enthalpy of the fluid moving through it. This is
carried out by either doing work on the fluid (e.g. compressor) or having work done on
it by the fluid (e.g. turbine), depending upon the effect required in the machine.
Therefore, the aerodynamic quality of the blades is of great importance.

The main objective of turbomachinery blade design is to realize a given velocity triangle
with minimal losses. Traditionally, empirical methods based on cascade test results have
been the basis of blade design. With the advent of computers, aerodynamic designs are
now based on the results provided by numerical simulation of the internal flow field.
The design process is an iterative one, requiring tools of increasing complexity as the
design develops. Early in the process, two-dimensional or quasi three-dimensional
inviscid flow analysis codes (e.g. those of Katsanis and McNally, 1977 and Hirsch and
Warzee, 1979) may suffice but in the final stages, advanced viscous three-dimensional
calculations (e.g. that of Dawes, 1988) need to be carried out.

The integration of these Computational Fluid Dynamics (CFD) methods into the design
system allows the designers to assess the quality of turbomachinery blades without
having to build prototypes in the early stages of the process. In most cases, the cascade
flows are highly complicated and three-dimensional. In high-speed cases, transonic or
supersonic flow may develop with complicated interactions between shock and
boundary layer. Despite the complexity, understanding of the flow phenomena has been
achieved to a high level; for example the effects of secondary flow (Kim et al., 1992),
the effects of tip leakage flow in transonic flow (Copenhaver et al., 1996, Suder and
Celestina, 1996) and the inception of rotating stall in unsteady flow (Ismael and He, 1997, Hoying et al., 1999), to name a few.

In the design problem, understanding of the fluid dynamics is directed at identifying areas where improvements can be made. For example, it is recognized that the strong acceleration of surface flow velocity on the suction surface of a blade is usually followed by strong diffusion leading to low energy flow further downstream. As a result, surface flow velocity must be carefully controlled to avoid high losses; Goldstein and Mager (1950) and Papailiou (1971), for example, proposed a roof-top type suction surface velocity distribution that has a rapid acceleration near the leading-edge followed by a plateau, and then a deceleration region that is pushed rearward to delay the transition to a turbulent boundary layer.

Despite the clarity of the design objective in this case, it is not clear how it may be achieved. The CFD calculation predicts the flow field of the current design only but does not give exact instructions on how the blade geometry may be modified to give the desired flow behaviour. In the conventional approach (also known as the "direct" approach), the blade shape is modified on a "trial and error" basis, guided only by empirical rules and/or the designers' experience. As a result, it can be laborious and difficult, especially if an unfamiliar design is encountered.

Modern turbomachines require more advanced design techniques than are adopted in the "direct" approach. A more efficient procedure for the definition of three-dimensional design is carried out by means of "inverse" methods, which allow for the generation of the blade geometries based on a prescribed target flow distribution.

This thesis is concerned with the development of two three-dimensional inverse procedures that can be applied to the systematic design of turbomachinery blades.
1.2 THE INVERSE DESIGN METHODOLOGY

Many inverse methodologies exist for the aerodynamic design of wing sections and turbomachinery cascades. The majority of these are two-dimensional and compute the blade shape based on prescription of the distribution of the surface static pressure or velocity. The surface distribution allows for good control over the flow-field; however, it is difficult to control the blade thickness distribution using this approach. Furthermore, this type of specification generally faces the problem of having no feasible solution and cannot be readily used in 3-D because of compatibility issues.

In the context of two-dimensional, incompressible and irrotational potential flow, constraints can be applied to ensure compatibility of the target pressure distributions in the design (Lighthill, 1945). Outside the theoretical limits, it has not been possible to guarantee any feasible geometry corresponding to a given target pressure distribution (Demeulenare, 1997). Three-dimensional designs are especially difficult since the radial relationship between the pressures on the hub and shroud has to be respected and the specification cannot be imposed independently. In the absence of a known geometry, the relation can only be approximated through guesswork and the program user must go through several numerical trials before finding an acceptable solution. Under these circumstances, some authors prefer to seek approximate solutions by relaxing the target distribution during the computation while others settle for a constrained or partial design where only one side (usually the suction surface) or a part of the blade is modified.

One design specification which has been applied successfully to the inverse design of blades in three dimensions without such difficulties, is the circumferentially averaged swirl velocity (or $r\bar{V}_\theta$) defined by,

$$r\bar{V}_\theta = \frac{N}{2\pi} \int_0^{2\pi} rV_\theta d\theta$$  \hspace{1cm} (1.1)

where $N$ is the number of blades in the cascade, $r$ is the radius and $V_\theta$ is the tangential velocity. The specification of $r\bar{V}_\theta$ directly satisfies the required specific work expected
of the blade (as given by the *Euler equation for turbomachinery*) and the parameter is related to the blade pressure loading, \( \Delta P \) and therefore, the pressure distribution. Hawthorne *et al.* (1984), Tan *et al.* (1984) and Zangeneh (1991) show that the pressure jump relation in the limit of a thin blade in incompressible, irrotational flow is given as,

\[
\Delta P = P^+ - P^- = \frac{2\pi}{N} \rho V_{m_{\theta}} \frac{\partial r \bar{V}_\theta}{\partial m}
\]

(1.2)

where the superscripts + and - denote the upper and lower surfaces of the blade respectively, \( V_{m_{\theta}} \) is the meridional velocity at the blade and \( \frac{\partial r \bar{V}_\theta}{\partial m} \) is the meridional derivative of the averaged swirl velocity. Although the specification of the averaged swirl velocity may seem to be less direct than the conventional surface pressure or velocity distribution in controlling the surface flow, there are advantages in using this as the design parameter.

For example, Borges (1986) showed that the exit loss in radial inflow turbines can be reduced by specifying a distribution of \( r \bar{V}_\theta \) which results in a zero span-wise variation of circulation and hence no shed vorticity. Zangeneh *et al.* (1996) proposed a versatile way of prescribing the target \( r \bar{V}_\theta \) distribution (see Appendix VII) and showed that by applying simple guidelines to the meridional derivative of the swirl velocity, \( \frac{\partial r \bar{V}_\theta}{\partial m} \), it is possible to adjust the pressure coefficient on the blade surface to minimize the secondary flow and reduce non-uniformity of the exit flow.

There are many other three-dimensional applications where the specification of the swirl velocity distribution has been found to be very useful. For example, in the case of aircraft propellers, the optimum \( r \bar{V}_\theta \) distribution has been found to reduce drag (Theodorsen, 1948). Designs of marine ducted propellers have also benefited from the specification of the \( r \bar{V}_\theta \) distribution, where optimization-based methods are set up with inverse design procedures to yield high performance propulsors (Black, 1997 and Yiu and Zangeneh, 1998a).
With the success and understanding achieved in the use of $rV_0$, this parameter was adopted by the author for developing a three-dimensional inverse design method applicable for the design of high-speed turbomachines.

It is also possible to develop another inverse design method based on the specification of the surface pressure difference distribution (equation 1.2). This method would complement the first by giving the user the option to carry out a design based on the flow aerodynamics along the blade surfaces. The development of this second method based on $\Delta P$ was therefore also pursued.

1.3 FLOW ANALYSIS

The first part of the current work involves selecting an appropriate flow solver. To accomplish all the design objectives (see section 1.4), the chosen method must be accurate in the prediction of flow in all regimes: subsonic, transonic and supersonic. In addition, the technique employed must be computationally efficient in order to yield an economically feasible design procedure. Different types of flow solutions were considered.

The most general equations are the time-dependent Navier-Stokes (N-S) equations, whose solution is valid for the entire spectrum of turbomachinery since the transport phenomena of friction, thermal conduction, and/or mass diffusion are included. Although Reynolds-averaged Navier-Stokes solutions are exact, with the exception of the modelling of transition and turbulence, the differencing scheme and the level of convergence, they are neither economical nor efficient; hence they are usually simplified to make them more attractive for use in many problems. These versions of the N-S equations usually neglect the stream-wise diffusion or normal viscous terms when they are thought to be physically insignificant in the flows concerned. Even so, the computational cost of solving the Navier-Stokes equations in a three-dimensional application remains high, particularly when it is used iteratively in a design cycle.
By contrast, solution of potential equations is very much easier and economical. However, the isentropic potential flow model has a limited range of validity, mainly in subsonic flow regimes. In transonic flow regimes, it has been found that the isentropic assumption in the flow solution leads to non-uniqueness of the transonic potential flows, resulting from the breakdown of the model with increasing shock strength (Hirsch, 1995).

Solution of the Euler equations gives the ability to model inviscid, compressible flow in two or three dimensions and in all regimes of flow. The Euler solver thus emerges as an acceptable compromise between solving the Navier-Stokes and the highly simplified potential equations. A more practical consideration is that most turbomachinery flows are determined primarily by inviscid effects such as flow turning, blade thickness and the blunt leading-edge of a lean and skewed blade, most of which can be adequately captured by Euler equations. In the cases where viscous effects dominate (for example, flow in a transonic compressor), approximation methods can be implemented with the Euler solver to model these effects. The development of such an Euler solver is therefore carried out. In particular, a time-marching finite volume solver of the unsteady Euler equations with viscous modelling is developed as the "black-box" that gives the steady-state flow solution for use in the inverse design procedure.
1.4 SCOPE AND OBJECTIVES OF CURRENT STUDY

The objective of the study is to develop inverse design procedures that are applicable to the design of turbomachines in an arbitrary configuration (axial, radial or mixed type). The methods should satisfy the following criteria:

1) Applicability in two or three dimensions.
2) Applicability in viscous, compressible flow, which can be subsonic, transonic or supersonic.
3) Ability to produce realistic blade geometries with finite thickness.

The thesis is organized to report the work carried out in order to achieve these objectives.

Chapter two outlines the current design practices, providing the context in which the present research is undertaken. Optimization methods are briefly described first. The chapter then focuses on the better-known inverse methods and summarizes their shortcomings in terms of their approach to seeking realistic 3-D blades. From the survey, it may be concluded that more rational inverse methods can be developed.

Chapter three contains the details for the development of the two- and three-dimensional analysis code, based on the solution of the Euler flow equations. The main constituents of the time-marching finite-volume solver and the modelling of viscous effects are described in this chapter.

Chapter four presents the validation of the developed solver. The verification of the computational results is carried out using published experimental data. Several practical test cases are used for this purpose; these include a transonic annular nozzle and the NASA rotor 67, both of which are recommended transonic test subjects given in AGARD report 275. The test cases presented here are later used for the validation and application of the two new inverse procedures in Chapters six and seven.
Chapter five gives the mathematical theory and formulations of the two design methods. The implementation of both design procedures and the derivation of their respective blade update algorithms are introduced.

Chapter six follows to validate the first design method based the specification of the mass-averaged swirl velocity, designated as Method I. In addition to the validations, two- and three-dimensional case studies demonstrate the working of this method and identify its limitations. The method is first demonstrated by redesigning a two-dimensional generic transonic compressor cascade, where particular attention is given to weaken the strong shock formation within the passage. Applications of the method in three dimensions involving redesigns of both the annular nozzle and the NASA rotor 67 are then presented.

In Chapter seven, the second design method, designated Method II, based on the specification of the distribution of blade pressure difference, $\Delta P$ is validated. Several designs of a generic two-dimensional transonic turbine cascade are performed to demonstrate the methodology. This is followed by the application of the method in three dimensions, where the NASA rotor 67 is redesigned. Some improvements in two problem areas are achieved. The performance of one of the new designs shows a qualitative improvement in the adiabatic efficiency compared with the original blade.

Chapter eight concludes the thesis with a summary of the studies made in the main chapters. This is followed by suggestions of further work that may be undertaken to improve and extend the current work.
CHAPTER 2
LITERATURE SURVEY

In Chapter 1, it was pointed out that the design of turbomachinery blades relies heavily on a variety of different CFD codes. A comprehensive review of the CFD techniques used as engineering tools in modern design system is given by Casey (1994). In this chapter, literature relevant to the present research is summarized.

Section 2.1 describes the conventional design approach to turbomachinery blade design. As the exact details of the design approach used in industry are often proprietary information, the material given in this section is just a brief outline.

To cope with increasing commercial pressures, the industry is beginning to pursue optimization techniques and inverse methods in order to help them produce high quality components in the shortest time. Although optimization is not in the scope of this thesis, it may be applied with inverse methods and hence, some techniques are described in section 2.2. This is followed by a more detailed review of existing inverse methodologies in section 2.3.

2.1 "CUT-AND-TRY" DESIGN METHOD

Classical methods of generating blade profiles have mainly been based on the superimposition of a prescribed thickness distribution on a circular or parabolic camber-line (Dunavant and Erwin, 1956, Havakechian and Greim, 1999). This method of profile generation is restrictive and local refinements of the blade shape are not easily achieved. More flexible alternatives have been the use of Bezier-Bernstein or B-Spline curve representation schemes. Two curves are used to describe separately the surfaces of the blade profile and by moving the control points, local geometrical changes can be achieved (see figure 2.1)
Figure 2.1: Illustration of Profile Shape Change by Moving the Bezier Points
(Reproduced from Havakechian and Greim, 1999)

The base profiles are optimized at a few radial positions and then stacked radially to give the 3-D blade. The way in which they are stacked can affect the performance substantially and this has been another area of research (Walker, 1987, Denton and Dawes, 1999). The resulting 3-D geometry is then checked using CFD codes.

There is no definitive rule dictating how the blade should be changed to give better performance. In fact, the basis of the method is primarily "cut-and-try", relying mainly on the intuition and experience of the designers, and perhaps following well-tried in-house practices, established from the design of many generations of similar machines.

The conventional approach has given rise to many high performance blades with advanced 3-D features; for example, end-bends in axial compressors (Wisler, 1984), compound blade lean in steam turbines (Hourmouziadis and Hubner, 1985), and swept leading- and trailing-edges in high-speed transonic compressors/fans (Wennnerstrom, 1984). However, such an approach is very time-consuming, labour intensive and subject to the creativity of the specialist designer.
2.2 OPTIMIZATION METHODS

Optimization techniques automate the design process by seeking a better design than a datum subject. Existing techniques in aeronautics and turbomachinery applications are mainly gradient-based methods or exploratory algorithms.

Optimization techniques are commonly applied and developed in the aeronautics industry where components of a flying vehicle are optimized (see review by Dulikravich, 1992); for example, isolated helicopter blade aerofoils (Reneaux and Thibert, 1985) and multi-component airfoils (Ormsbee and Chen, 1972).

The structure of the control of optimization problems contains three main components (Gunzburger, 1997). The first are the objectives, which are what one hopes to achieve by controlling the system. These may be to minimize the loss; or for aerofoils, to maximize the lift, to minimize the drag and so on. There must also be controls or design parameters that describe how the system is to be adjusted; and finally, there must be constraints that describe the system that is to be controlled.

Figure 2.2: Geometry Model: Basic Parameters and Bezier Curves
(Reproduced from Pierret, 1997)
In the aerodynamic shape design of turbomachinery blades, controls may be provided by B-spline or Bezier curves to give advanced definition of the blade, for example that of Pierret (1997) as shown in figure 2.2.

Of the different classes of optimization techniques, *gradient-based* methods (GM) have been the most widely used. Here, the sensitivity of the profile losses to small perturbations of the Bezier curve is established and then used systematically to adjust the geometry until the losses are minimized. The formulation of the method is relatively straightforward relying mainly on establishing and calculating the gradients of the function (i.e. the loss) to the constraint (the geometry shape). However, the technique is not robust (Obayashi, 1997) especially in blade design problems where the objectives and constraints (or functional) are not necessarily differentiable or convex. GM methods are also highly dependent on the initial points (i.e. the initial geometry) and can get stuck in local extrema, unable to locate the required global optimum.

*Exploratory techniques* avoid focusing only on a local region and their stochastic natures generally evaluate designs throughout the parameter space in search of the global optimum. *Genetic algorithms* (GA) form one class of such techniques. They are known to be robust as they do not require the functional to be well-behaved. However, these techniques, which are essentially search algorithms based on the mechanics of natural selection in the evolution of populations, are highly demanding in terms of computational requirement and are often too expensive when applied to shape design. Shelton *et al.* (1993), for example, formulated the method for the optimization of transonic turbine blades but rejected the method due to its high computational cost, in favour of hill-climbing (gradient-based) techniques. A similar conclusion was reached by Goel *et al.* (1996), who compared several optimization techniques in order to implement a feasible automatic optimization tool for blade profile design. One exception is the method of Trigg *et al.* (1999), which is based on a genetic optimizer coupled with a Navier-Stokes solver to minimize profile loss in 2-D blade profiles. Trigg was reported to have achieved a 20% improvement in the profile loss of a 1960's steam turbine blade profile, in 10% of the time needed had the work been performed without the optimizer. The superiority of Trigg's GA method over previous attempts
may be attributed to the more advanced models used for the population manipulation and selection processes. These increase the efficiency of the selected population in reacting and recombining (i.e. mutating) to produce the "fittest" profile.

Another exploratory technique is the Simulated annealing (SA) method. This method is beginning to appear in aerodynamic shape optimization problem. The term "annealing" refers to the process in which a solid metal is first heated and then allowed to cool by slowly reducing the temperature. If the metal is cooled too quickly (i.e. quenched), it will not reach the global minimum state of its potential energy state. Based on this principle (Metropolis et al., 1953), SA has been very attractive for large scale optimization problems, especially where the desired global extremum is hidden among many, poorer, local extrema (Press et al., 1992). A recent study by Pierett and Van den Braembussche (1998) has explored the use of Artificial neutral networks (ANN) vsdth SA as the optimizer for the design of turbomachinery blades. The combination of the "learning" ability of ANN and the stochastic nature of SA gives rise to what may be a powerful knowledge-based design system.

In recent years, there has emerged an alternative to direct optimization. Instead of iterating directly the geometry, inverse optimization methods deal with a design parameter (e.g. pressure distributions). Once the target pressure distributions are optimized, the corresponding blade shapes are then determined by the inverse methods. Obayashi and Takanashi (1996) implemented such a procedure where the inverse method of Takanashi (1985) is used to alleviate the computational time of a standard GA procedure. Very remarkable optimum designs of aerofoil and wing designs were achieved with their method. An automatic design procedure was also developed by Yiu and Zangeneh (1998b) for the design of optimum impeller blades. In this method, the circumferentially averaged swirl velocity, \( r\bar{V}_\theta \) is parametrized by a cubic B-spline using control points, and a simple line search method is invoked to minimize the loss. This line of research has also been pursued by the author (see Tiow et al., 2000), using the Simulated annealing algorithm with one of the proposed inverse design methods described in this thesis (Method I, Chapter 6).
2.3 **INVERSE METHODS**

Inverse methods, as their name suggests, involve working in the reverse direction from that originally adopted in the conventional or direct design approach. As mentioned earlier, the cut-and-try direct approach in turbomachinery blade design suffers from the inefficiency of repeated analysis of the current design and of modifying the geometry based on empirical rules or the designers' own experience.

The inverse (or design) approach is more efficient, allowing for direct computation of the geometries based on a prescribed performance. There is a diverse selection of methods, some of which are now routinely used as part of the design process in establishments with CFD fully integrated into their system.

The following sections present an overview of some of the better-known inverse methods, classified by the flow equations on which they are based. The reviews outline the methodologies and identify their abilities or shortcomings in the application to turbomachinery blade design. A summary of the methods is shown schematically in figure 2.5, at the end of this chapter.

### 2.3.1 Methods Based on Potential Flow Equations

The first generation of inverse design techniques is based on the potential flow equations. The inverse problems in this category are solved either analytically or iteratively. The plane transformation method is one broad class of inverse design methods in which the shape geometry is determined analytically. Sections 2.3.1.1, 2.3.1.2 and 2.3.1.3 describe a few examples of methods in this category.

Other techniques formulated in a pure mathematical framework are also reported. One popular technique is the *fictitious gas* approach, which has been applied to designing shock-free transonic configurations. This is discussed in section 2.3.1.4.
The inverse problem can also be solved iteratively. One such approach employs the method of singularity, which includes the widely-used Circulation method. This is discussed in section 2.3.1.6.

2.3.1.1 Conformal Mapping Methods

Lighthill (1945) was perhaps the first to resolve the mathematical difficulties associated with inverse aerofoil design. In his pioneering work, conformal mapping, commonly used for classical aerodynamic analysis of flow around arbitrary shapes in incompressible flow regimes, was applied successfully to calculate the aerofoil geometry based on a target surface velocity distribution.

![Conformal Mapping](image)

**Figure 2.3: Conformal Mapping**

In this approach, the invariant mapping is carried out using a Laplace operator with the complex plane containing the aerofoil transformed into a unit circle, where the target velocity is imposed. Special guidelines were formulated by Lighthill to constrain the specification of the target velocity in order to achieve a meaningful solution of the aerofoil geometry. These constraints include conditions to ensure closure of the aerofoil profile and compatibility of the target velocity distribution with the far field condition.

The original technique is restricted to incompressible flow regimes. In the compressible regime when the full potential equation is solved, the mapping is no longer invariant and thus both the implementation and the constraints have to be modified. Woods (1952) accounted for compressible flows using the Karman-Tsien approximation and derived the constraints for selected cases. Volpe and Melnik (1986) also proposed a version for
the design of aerofoils in compressible flow, satisfying the constraints numerically by introducing free parameters and predefined shape functions in the target velocity distribution. These parameters are adjusted during the computation to guarantee the existence of a solution as well as closure of the aerofoil's trailing-edge. Although this ensures that the inverse problem remains well-posed, the technique is approximate and it is difficult to achieve the target velocity closely.

All the methods reported thus far are limited to two-dimensional single aerofoils and their formulation is restricted to irrotational potential flow, where conformal mapping and its analytic properties can be applied or approximated. The inherent mathematical assumptions and restriction to two dimensions are too severe for practical use in the design of turbomachinery blades.

2.3.1.2 Potential and Stream Function Methods

Another approach employing plane transformation tackles the inverse problem in the popular potential-stream function plane \((\phi, \psi)\). This approach was first proposed by Stanitz (1953) for the design of channels and cascades of high solidity, in which the velocity is prescribed as a function of the surface length in the physical \((x, y)\) plane. In the transformed \((\phi, \psi)\) plane, the irrotationality and continuity equations are solved together with the prescribed boundary conditions to give the steady, inviscid and irrotational flow field. Having determined the flow solution, the flow angle is then integrated in a back transformation to recover the channel shape between the blades in the physical \((x, y)\) plane.

Stanitz (1980) extended the method to three dimensions, introducing a second stream function and solving the problem in the \((\phi, \psi_1, \psi_2)\) plane. Stanitz demonstrated the ability of his method in a range of three-dimensional channels and ducts restricted to low speed subsonic flow.
Schmidt (1980), Schmidt and Berger (1986) and then Bonataki et al. (1993) adopted Stanitz’s method for the calculation of turbomachinery blade cascades. The assumptions of a steady and irrotational two-dimensional flow were still necessary and the methods were restricted to shock-free applications.

Schmidt modified the technique to compute for supercritical compressor and turbine cascades in transonic flow, but the technique had to employ distinct finite difference equations to cope with different domains of dependence (i.e. elliptic, parabolic and hyperbolic) in the flow field, depending on whether the region was subsonic, sonic or supersonic. Schmidt and Berger (1986) presented a detailed account of the necessary implementation. The resulting method is complicated and requires meticulous effort to identify different flow regimes in a transonic cascade computation. In the local supersonic region, the prescription of the velocity distribution becomes ill-posed and is thus physically meaningless. In such cases, the numerical solution would not converge. In addition, an incompatibility was encountered between the surface velocity distributions on the pressure and suction sides of profile, giving rise to non-feasible profile shapes with open ends.

Bonataki et al. (1993) followed the work of Schmidt and extended the application to include the design of quasi-three-dimensional turbomachinery cascades. Bonataki addressed the problem of having a profile with open or cross-over trailing-edge by changing the solidity and the chord of the cascade iteratively in the computation until a closed blade section is obtained. This is a reasonable step towards tackling the problem but it compromises the control on the cascade configurations (for example, having to change the number of blades in the cascade during the computation).
2.3.1.3 Hodograph Methods

In this approach, the inverse problem is solved on the hodograph plane where the physical flow is represented in terms of the resultant velocity, \( V \) and the streamline angle, \( \theta \). The main advantage of the hodograph method is that it simplifies the non-linear full potential equations into a linear system in the new coordinates \((V, \theta)\).

The analytical hodograph design method was first developed by Nieuwland (1967) and later adopted by Boerstoel and Huizing (1974) and Bauer, Garabedian and Korn (1972, 1975, 1977). Nieuwland's method was used to calculate a family of quasi-elliptical aerofoils. Boerstoel and Huizing provided shock-free supercritical aerofoils using a linear combination of simple compressible flow solutions for the two-sheeted hodograph surface of lifting flows. The method of Bauer, Garabedian and Korn was more general and was also demonstrated in the design of highly cambered shock-free aerofoils. This method was very successful in defining a class of useful supercritical aerofoils and is documented in some specialized texts (e.g. Schrier, 1982, Lakshiminarayana, 1996). Despite its success, their method is two-dimensional and practical use of the method requires much experience and mathematical insight (Yiu, 1994).

In the approach of Sobieczky (1979) and Hassan et al. (1984), the hodograph variables are chosen to be the Prandtl-Meyer function, \( \nu \) (which depend only on the local Mach number) and the flow angle, \( \theta \). The system of equations is not solved directly in the hodograph plane; instead, a conformal transformation is employed to further transform the closed curve representing the aerofoil in the hodograph plane into a unit circle. In the circle plane, the target pressure (or Mach number) distribution is readily applied and the system is solved to give the corresponding \( \nu \) and \( \theta \). In conclusion, the final aerofoil shape is defined using the solution when it is transformed back to the physical domain.

In the design of shock-free supercritical aerofoils and cascades, the subsonic region of flow is decoupled from the supersonic region. The part of the blade profile in the supersonic region is solved using the solution in the subsonic region as the initial data.
for the initial value problem and the supersonic part of the profile is moved to make the flow shock-free.

The main problem in this class of methods is that the results are not easily grasped by the engineer due to the use of a velocity coordinate. More serious shortcomings are that the techniques are restricted to shock-free, two-dimensional calculations and the user does not always have full control of the pressure distribution. As a result, this category of methods is classified by some to be "indirect" as opposed to "inverse", since in this case, the designer has control over neither the aerodynamic quantities nor the geometry (Van Den Dam et al., 1990)

2.3.1.4 Fictitious Gas Methods

The fictitious gas technique, like the hodograph method, is formulated to design a shock-free flow field. It is known more specifically as the Sobieczky fictitious-gas elliptic continuation technique (Sobieczky et al., 1978) or as the Nakamura gas model (Nakamura, 1981), and is mostly used for producing shock-free transonic wing profiles.

The design concept is based on the fact that shocks can form only if there is a supersonic flow; that is, if the governing partial differential equation is locally hyperbolic in nature. Consequently if shock-free flow is desired, the partial differential equation should never be hyperbolic. Sobieczky and Nakamura achieved this by modifying the local density to be equal to the critical density whenever the flow becomes supersonic.

In the subsonic region, including the sonic line that forms the boundary with the supersonic region, the solution obtained is based on the correct governing equations. The solution for the supersonic region based on the fictitious gas concept is, however, non-realistic and is thus discarded. Recalculation of the solution within the isolated supersonic bubble is then carried out based on the correct gas law using the solution along the sonic line as initial data. The newly-determined surface streamline of the real
gas flow within the supersonic bubble becomes the new profile segment and the procedure is then iterated until the new coordinates converge.

The approach is neat in that only the portion of the aerofoil in the supersonic regime is modified in order to achieve a shock-free flow. Three-dimensional shock-free wing designs were reported using this method (Yu, 1980 and Fung et al., 1980), although the cases are restricted to situations in which there is only a small region of supersonic flow.

Dulikravich and Sobieczky (1981) adopted the technique for the shockless design of two-dimensional transonic turbomachinery cascades. The computed design does not suffer from the problem of open-ended profile shapes but, like the hodograph method, the user does not have control over the flow field or the geometry. As a result, it is also classified as an "indirect" method.

2.3.1.5 Taylor Series Methods

Wu and Brown (1952) pioneered this approach in which they proposed that series expansion methods could be performed on selected streamlines to design two-dimensional blade profiles. The applicability of this approach to the design of turbomachines was then recognised by several researchers and its development was continued mainly in China, as reported by Cai (1984).

Zhao et al. (1985) extended the method to three-dimensional design of turbomachinery blades and the inverse problem was set up in the three-dimensional domain consisting of two intersecting families of relative stream-surface, the $S_1$ and $S_2$ planes. In the computation, the hub and casing wall contours and the projection of the mid-passage stream-surface ($S_{2m}$) are specified together with a fixed blade circumferential thickness and the distribution of $rV_\theta$ along this surface, $r$ being the radius and $V_\theta$ the tangential velocity.
The \( S_{2m} \) flow gives the shape of the \( S_{2m} \) surface, and a Taylor series expansion is then used with the irrotationality and continuity equations to give the shapes of the family of \( S_2 \) stream-surfaces progressing from \( S_{2m} \). The extreme two surfaces of the family of \( S_2 \) surfaces are then the upper and lower surfaces of the cascade. Wang (1988) also reported a similar formulation of the method applied to the design of three-dimensional axial flow blades.

This approach is reported to be highly efficient (Dulikravich, 1990) but the flow solution is only approximate as accuracy is limited by the number of terms in the series expansion. Usually only the first few terms are included; for example, Zhao included only the first three terms in his scheme while Wang used up to five terms for higher accuracy. In addition, the approximation does not cope well in regions where the flow experiences abrupt changes; it is noted that errors in the computation increase rapidly towards the stagnation points and the results are only accurate for blades with high solidity (chord to pitch ratio).

### 2.3.1.6 Singularity Methods

In this approach, the flow field and the blade shape are represented by a distribution of sources, sinks and vortices. Goldstein and Jerison (1947) proposed the singularity method in designs based on a prescribed velocity distribution. This was followed by Schwering (1970) who used this approach for the two-dimensional inverse design of blade cascades in which the flow is assumed to be incompressible and made up of the superposition of a uniform flow and a circulation flow.

Along the blade contours, the flow function is reduced to a Fredholm integral equation of the first kind for the circulation distribution, which is identical to the known velocity distribution. The equation is then integrated to give the profile contours assuming that they coincide with the surface streamlines.
Betz and Flugge-Lotz (1939) also used the method of singularities for the design of two-dimensional radial turbomachines. In this approach, the inverse problem is formulated in polar coordinates \((r, \theta)\) and the cascades are assumed to be thin. The vorticity is represented by a periodic delta function and assumed to be concentrated on the blade surfaces while the flow remains irrotational everywhere else. The circulation is specified along the radius and the shape is calculated iteratively.

Hawthorne et al. (1984) reported a similar method applied to the design of two-dimensional thin blades in rectilinear cascades. In his method, the blades are represented by a distributed bound vorticity, whose strength is given by the prescribed pitch-averaged tangential velocity, \(\bar{V}_\theta\). Hawthorne presented two methods of solving the inverse problem, one based on the classical method used by Betz and Flugge-Lotz and the other based on the Clebsh formulation of steady rotational flows. The superiority of the latter method was demonstrated in concurrent work by Tan et al. (1984) where the three-dimensional version of Hawthorne's method was given. The three-dimensional method designs blades for a prescribed distribution of circumferentially averaged swirl velocity, \(r\bar{V}_\theta\). A similar method was also given by Borges (1986, 1993). In these methods, the velocity field resulting from the computed vorticity distribution is used to determine the blade shape. This determination is carried out by integrating the tangency condition (i.e. the inviscid slip) from a fixed point of the blade, for example at the leading-edge to the other end.

Further developments were reported by Zangeneh (1991), who accounted for flow compressibility and approximated the effect of blade thickness based on the prescription of some blockage factor. The idea of blockage is demonstrated to be adequate in the modelling of small thickness, typically encountered in radial turbomachinery blades.

The methods initiated by Hawthorne are now more commonly referred to as **Circulation methods** and are some of the earliest methods that could be formulated and applied successfully in full three dimensions. Through these methods, it is also shown that the specification of \(r\bar{V}_\theta\) can be used with good effect to improve three-dimensional blades.
in subsonic flow, for example radial inflow turbine blades (see Zangeneh, 1988). This specification is thus a useful alternative to the more conventional choice of surface pressure or velocity distribution.

2.3.1.7 Another Method Based on the Specification of $r\vec{V}_\theta$

In an effort to develop a three-dimensional method based on the specification of $r\vec{V}_\theta$, Soulis (1985) proposed an iterative procedure in which the blade shape is adjusted directly from the difference between the calculated and specified $r\vec{V}_\theta$ distributions.

In this method, the flow result is provided by a direct finite-volume flow solver, which solves the three-dimensional, inviscid, isentropic and potential flow through a turbomachinery blade row. Following the flow computation, the blade is modified by satisfying the surface flow tangency condition.

Soulis demonstrated the capability of the procedure in the design of three-dimensional free-vortex turbomachinery blades in subsonic compressible flow. In all the cases represented, the blade is assumed to be stationary and to have zero thickness.
2.3.2 Methods Based on Euler Equations

The next broad category of inverse design methods is based on the solution of the Euler equations of motion. Although the methods are based on inviscid equations, the flow is rotational and the methods are applicable to design problems in all regimes of flow; subsonic, transonic or supersonic. In situations when shocked flows are encountered, accurate shock capturing schemes are usually in place to model the phenomena and the entropy generation is properly accounted for, since the flow is not assumed to be isentropic.

The methods in this category are mainly iterative and consisting of two main parts. The first is the Euler flow solver (usually modified, for example using a permeable wall) and the second is a suitable blade update algorithm.

The majority of the Euler-based methodologies prescribe the surface pressure or velocity distribution. The required quantities are imposed directly on the surface of the present geometry and if the distribution is different from the one that would result from a direct analysis of the present geometry, the surface velocity field would not be aligned with the walls. The geometry must then be adjusted to correct the situation. There are two main approaches to the idea, the first being to assume that the walls are not fixed but are allowed to move with the flow; while the second is to assume that the walls are fixed but the surfaces are permeable such that normal transpiring velocities can go through them, which are then used to determine the new geometry. The latter approach is also known as the Transpiration model and is discussed in more detail in section 2.3.2.2.

2.3.2.1 Movable and Impermeable Surface Model

In this approach, the surface velocity or static pressure distribution is prescribed to give the required geometry of an aerofoil. The required quantities are imposed as a boundary condition along the impermeable wall of the profile and after each iteration, the grid
lines defining the boundary of the aerofoil are moved so that they are aligned with the computed surface velocities. The geometrical adjustments are computed by integrating the local slopes of the velocity. This process is iteratively pursued until the flow analysis converges to the steady solution and the displacements of the mesh points between two successive iterations become negligible.

Meauze (1982) developed such a method and applied it to two-dimensional turbomachinery cascades and duct design using solutions of the unsteady Euler equation. As in the previous methods which prescribed distribution of pressure or velocity on both the upper and lower sides of the blade profile, Meauze encountered problems of non-converging cases and the occurrence of non-physical blade profiles with open trailing-edges and fish-tail shapes. In such cases, Meauze proposed a few options,

1) Modifications are made only on the suction side of the blade and a fixed thickness is specified. The resulting pressure side is determined by subtracting the fixed thickness from the modified suction side;
2) The cascade solidity is not assigned but is varied during the computation such that the trailing-edge of the profile always remains closed; and,
3) In the case where the cascade solidity is assigned, successive alterations of the prescribed pressure distribution are made during the computation if no realistic geometry exists from the original prescription.

Subject to the compromises made in options 1-3, Meauze demonstrated the capability of the method in the design of several turbines and transonic compressor blade profiles. Zannetti (1980, 1987) developed a similar method and applied it to the design of fully three-dimensional turbomachinery blades. Encountering the same difficulties, Zannetti adopted the constrained design approach (similar to option 1 of Meauze's suggestion) where only the suction side of the blade is designed.

The methodology proposed by Giles and Drela (1987) is based on the simultaneous solution of multiple stream-tubes, coupled through the position of, and pressure at, the
streamline interfaces. Their idea of replacing the wall surface with imposed pressure is similar to that of Meauze and Zannetti but they solved the discrete two-dimensional steady Euler equations in implicit form with the Newton method.

2.3.2.2 Surface Pressure Transpiration Model

In this approach, the direct solver, which usually carries out the analysis of the flow field for a known geometry, is transformed into a design method. This is done by replacing the classical inviscid slip condition with the imposition of the required static pressure distribution along the blade surfaces. By doing so, the wall boundaries become permeable (or porous) and allow fluid flow normal to the surfaces.

The basic idea of the transpiration model is that if the blade shape is not compatible with the required pressure distribution, the surface flow will not be tangential to the boundary of the geometry (i.e. the velocity component normal to the surface is non-zero) and the new shape can then be determined by resetting the current geometry such that its surfaces become parallel with the resulting transpiring surface flow velocity.

The basic idea is thus similar to that discussed in section 2.3.2.1, except that in this case flow analysis is carried out until convergence. The blade surfaces, although modelled as permeable, are fixed throughout the analysis.

The iterative process of analysis (based on the transpiration model) and blade update is continued until the normal surface flow velocity eventually diminishes, indicating that the blade shape has satisfied the required surface pressure (or velocity) distribution. Leonard (1990), Leonard and Van den Braembussche (1991) and also Pommel (1994) formulated the two-dimensional transpiration model and implemented the design method using a finite-volume, time-marching procedure to solve the unsteady Euler equations for compressible flow in the blade-to-blade plane. The cell-vertex approach, where the flow variables are stored at the corners of the cell, is adopted in the solver code.
The capabilities of the two-dimensional method were demonstrated in the design of both subsonic and transonic cascades. A three-dimensional extension of the method was later reported by Demeulemaere and Van den Braembussche (1996) who also applied it to the design of both compressor and turbine blades, which include examples of shock-free designs. In the three-dimensional version, the solver code uses the cell-centre approach where the flow variables are stored at the centroid of the cell; however, for accuracy in the computation along the surfaces, additional nodes are introduced at the boundary.

In all the reviewed works, there seem to be particular difficulties in specifying pressure distributions on the upper and lower side of the blade to give physically realistic designs, or any solution at all. In the former case, physically unrealistic geometry having either open ends or a fish-tail shape is encountered (see figure 2.4).

In these cases, a common remedy (Leonard, 1990) is to rotate the upper and lower surfaces about the leading-edge until the ends meet. This gives a physically realistic profile but means that the resulting blades do not satisfy the imposed pressure distributions.

It is known that the design pressure distributions must be checked to be compatible with upstream and downstream conditions of the cascade to result in a solution, and that arbitrary prescription of suction and pressure side distributions does not always bring convergence to a final blade shape. Unfortunately, there are no definite guidelines/constraints that can be derived with the flow predicted by Euler solutions.
In such cases, Leonard suggests a compromise be made in the design by relaxing the required distribution depending on the level of partial convergence attained. This strategy ensures that a solution which is at least near to the original specification is given. Pommel (1994) chose to avoid the problem by designing only the suction side of the blade while imposing a finite thickness distribution. Pommel argued that the approach is a reasonable implementation since the suction side is more critical in blade performance. Nevertheless, it is obvious that this is a constrained blade design approach where only one blade surface can be optimized.

2.3.2.3 Euler-Based Circulation Method

In this approach, Dang and Isgro (1994, 1995) reformulated the Circulation method using conservative variables for use with the Euler equations. As in the original Circulation method, the blade tangential or swirl velocity distribution is specified but discrete body forces in the form of bound vortex sheets are used to represent the presence of the blade. Like the earlier methods of Tan et al. (1984) and Zangeneh (1991), this Euler-based method is three-dimensional but the application is still restricted to thin blades (see section 2.3.1.6). Specifying the circulation distribution in an Euler solver in order to design a blade with finite thickness is difficult since it relies on an analytical relationship between the circulation and loading (related to the specified tangential velocity) to find the blade pressures required by the solver.

2.3.2.4 Methods Based on ΔP

A new transpiration model based on the surface pressure difference (ΔP) distribution was reported by Dang (1995). The new transpiration model was given in two dimensions and the flow calculations are performed using a finite-volume, cell-centre time-marching scheme to solve the unsteady Euler equations in the blade-to-blade plane. The design procedure is applied to transonic cascade designs subject to a ΔP distribution and a fixed blade thickness distribution, which can be chosen according to
mechanical requirements. Structural integrity of the design can therefore be always ensured. However, the details given by Dang show that the value of the flow variables along the boundary of the profile are assumed to be equal to that at the upper and lowermost cells adjacent to the surfaces. This is a necessary approximation since no flow properties are given exactly on the wall in a cell-centre formulation when the flow variables are kept at the centroid of the cell.

To compute the transpiring velocities, the flow variables in these cells are adjusted according to the specified pressure difference quantities. Another approximation is made here in which the densities and axial velocity components on the pressure and suction sides of the profile are assumed to be identical at a given location along the blade. Based on this assumption, the fluxes at the cell faces adjacent to the wall are evaluated and used to update the profile geometry in relation to the tangency flow condition. No further details are given regarding the validity of this assumption, although Dang used the flow computed in this manner as the flow field of the final geometry. Recently, Damle and Dang (1998) reported the three-dimensional application of the method to the design of the first rotor of a transonic stage.

Apart from the present author, a separate effort was also reported by Ahmadi and Ghaly (1997), who developed a two-dimensional version of the method using unstructured grids. In both methods, no assumption of the flow being identical on both sides of the blade is made and the surface flows are evaluated exactly on the surface boundary using a cell-vertex formulation.

There is one shortcoming of using $\Delta P$ as the design parameter. The difficulty is that the distribution cannot be specified arbitrarily to give the required specific work or turning. In the current work (Tiow and Zangeneh, 1998), an automatic procedure to seek the $\Delta P$ distribution that would give the required specific work is proposed. In addition, the computation was improved by the inclusion of viscous effects.
2.3.3 Methods Based on Navier-Stokes Equations

In recent years, there has been some interest in developing a procedure based on the solution of the Navier-Stokes (N-S) equations. Some examples are the methods proposed by Demeulenaere et al. (1997), Leonard and Demeulenaere (1997), Wang and Dulikravich (1995) and Wang et al. (1998).

The 2-D method of Demeulenaere is an extension of his earlier Euler-based method using the transpiration model (section 2.3.2.2). The design procedure is similar to the Euler-based design procedure except that the solution of the N-S equations uses a mixed explicit and implicit time-marching scheme. Only a small modification in the implementation of the transpiration model is required, which essentially eliminates the surface velocity terms in the original formulation. The boundary condition is then compatible with that in a direct analysis, which imposes zero velocity and normal pressure gradient at the wall.

Wang and Dulikravich (1995) employed a similar approach for the design of aerofoils. In this case, the Navier-Stokes equations are solved using an explicit time-marching approach with the Baldwin-Lomax model to account for turbulence flow. An extension of the method for the design of three-dimensional turbomachinery blades was also reported (Wang et al., 1998).

These methods are still limited by their computational cost and are more suitable for the final design stage when only minor adjustments of the geometry are required.
2.4 SUMMARY OF REVIEWS

In the previous sections, some of the existing techniques available to solve the aerodynamic inverse problem have been reviewed. Although by no means comprehensive, this review shows ongoing efforts to develop or improve inverse methodology for aerodynamic design. Here the main observations are summarized and figure 2.5 depicts the main points.

Different inverse design methods based on potential flow were described in section 2.3.1. In general, these methods are highly efficient and only a few iterations are required to compute the desired blade shape. However, this is only possible because assumptions are applied to simplify the flow equations. At the simplest level, the flow is assumed to be isentropic, irrotational and incompressible (for example, Lighthill's conformal mapping technique). Most of the methods in this category are also not fully three-dimensional except those of Yu (1980) and Fung (1980) and those of Tan et al. (1984), Borges (1986), Zangeneh (1991) and Soulis (1985).

The methods by Yu and Fung are based on the Fictitious gas model which do not allow the user to have full control over the flow specification or the geometry. Their methods thus fall short of a systematic design methodology. The second group of methods by Tan, Borges, Zangeneh and Soulis based on the specification of the swirl velocity is systematic and effective but the applications are limited to the design of thin blades.

In addition to the above, a common limitation in all the surveyed methods in this category is that they cannot cope with high-speed flow where there is shock. The methods may therefore be applied only in cases when the flow is entirely subsonic. Although the two-dimensional method proposed by Schmidt (1980, 1986) is adequate when the shock waves are weak, the rotational effects are neglected. The assumption of irrotationality is not acceptable in high-speed turbomachinery since the flow is rotational downstream of any shock (including weak shocks). Not only is this important to the description of the flow field around shock waves, but rotational effects are also important for accurately predicting real fluid behaviour and quantifying the actual force.
on the blade during the working of the fluid machine. The methods in this category are thus not applicable for the design of high-speed transonic turbomachinery blades.

Section 2.3.2 described some Euler-based methods. The primary advantage of Euler-based inverse methods over those based on potential flow is that they are applicable for all flow regimes including high transonic flow cases where there is strong shock. In addition, the flow model behind the design is more realistic since the flow is not assumed to be isentropic or irrotational.

Substantial effort in the use of the surface pressure distribution as the design specification has been reported. All such procedures including that using a Transpiration model suffer from the difficulty of obtaining realistic blades if the pressure distributions are specified on both the pressure and suction sides of the blade. As a result, a compromise is usually made to yield an acceptable result. Common approaches are to design only one side of the blade or to relax the design specification. More seriously, the prescription of this design distribution in three dimensions is extremely difficult since the distribution at the hub and shroud cannot be specified independently and it is not possible to know their relationship in the absence of a known blade geometry. This is a major limitation of the method.

An alternative to the surface pressure transpiration model method is to impose the blade pressure difference distribution together with a chosen thickness distribution. As this approach is in the early stages of development, only details in two dimensions have been reported and improvements are possible. The applicability of the design specification in three dimensions and its ability to design blades with finite thickness also make it attractive for further development. This is therefore pursued as one part of the current work.

The other part of the current work involves implementing another Euler-based method. From the successes reported in the use of the swirl velocity in sections 2.3.1.6 and 2.3.1.7, this is chosen to be design parameter for the method.
In recent years, there has been interest in developing design methods based on viscous flow solutions and a few methods employing Navier-Stokes equations have emerged. Due to the high computational requirement involved, however, these methods are still too expensive for routine use. In contrast, in the current work, viscous modelling is implemented to improve the Euler solution. The viscous coupling is efficient and allows designs to be carried out while taking into consideration the viscous effects, without incurring a severe increase in computational cost. Details of the inviscid/viscous solver code and the design methods are given in chapters 3 and 5 respectively.
Figure 2.5: Summary of Existing Inverse Methods
CHAPTER 3
DEVELOPMENT OF THE TIME-MARCHING EULER SOLVER

The finite-volume time-marching solver of the unsteady Euler equations of motion is an integral part of the proposed inverse design procedures. The steady state flow solution determined by the solver is used directly by the design methods (with some modifications to the wall boundary condition for the second method, details in chapter 5) to compute for the desired blade shape.

For this purpose, both two- and three-dimensional versions of the solver are developed. In this chapter, the main constituents of the solver code are described.

3.1 INTRODUCTION

There are several numerical techniques with which the solution of the Euler equations of motion can be sought. The problem is usually solved in practice through methods of the unsteady type. The time derivatives in the unsteady Euler equations make the system of equations hyperbolic in nature and the calculation is carried out iteratively to seek the flow solution whose steady state is considered to be the asymptotic state of an unsteady motion.
These time-iterative or time-marching techniques are commonly used and have the following advantages,

- *The same code is used for the solution of all flow regimes, from subsonic to supersonic.*
- *The technique is easy to implement and vectorize. A coupled system of equations allows pressure, density, velocities, enthalpy and temperature to be solved simultaneously.*
- *The code/technique is flexible and the same code can be employed for a range of internal problems like 2-D cascades, 3-D blade rows, 3-D unsteady rotors, multi-stage turbomachinery flows and 3-D unsteady rotor and stator interactions.*
- *Extensive development work has been performed on these techniques, and hence they are mature, robust and reliable.*

By employing these explicit techniques to solve the unsteady equations, steady state or time-accurate solutions can be achieved readily and the implementation of boundary conditions is straightforward.

Here, the Euler solver is developed using a cell-vertex approach (Ni, 1981) to compute the steady-state solution. A central difference scheme with added artificial viscosity and a Runge-Kutta integrator proposed by Jameson et al. (1981) are used as the numerical scheme to carry out the iterations. Appropriate boundary conditions are applied to represent the blade walls, the periodic planes and the inlet and outlet boundaries of the cascade flow problem. To accelerate convergence, local time-stepping and grid sequencing are implemented in the present computer code.

The following sections describe these main parts of the two- and three-dimensional solver codes. The equations are presented in three dimensions but are readily simplified to give the corresponding form in two dimensions.
3.2 THE GOVERNING EQUATIONS AND UNDERLYING ASSUMPTIONS

The Euler equations are a direct result of the application of Newton's second law of motion, along with the principles of conservation of mass (i.e. continuity) and energy.

The developed solver works with absolute flow quantities and applies the conservation equations to a fixed grid. The original Euler equations assume that the flow is inviscid, neglecting dissipative transport phenomena of viscosity and mass diffusion.

In the current developed two- and three-dimensional solvers, viscous effects are included using simple viscous models (more details are given in section 3.7). Thus, the only underlying assumptions in the current solution are,

1) *The flow medium behaves like an ideal gas.*

2) *There is no heat conduction in the medium.*

The effect of rotating blades is also modelled, and in such cases the grid coincides with the rotating domain. The implementation follows closely the work of Denton and Singh (1979).

In three dimensions, the system of equations for compressible flow cast in absolute cylindrical polar coordinates \((r, \theta, z)\) and written in integral form is as follows,

\[
\frac{\partial}{\partial t} \iiint_{\Omega} H dVol + \oint_{\partial \Omega} \left[ E \hat{n}_z + (F - Hr \omega) \hat{n}_\theta + G \hat{n}_r \right] dA = \iiint_{\Omega} (S_t + S_{\text{vol}}) dVol \tag{3.1}
\]

where

\[
H = \begin{bmatrix}
\rho \\
\rho V_z \\
\rho V_\theta \\
\rho V_r \\
\rho e
\end{bmatrix}, \quad E = \begin{bmatrix}
\rho V_z \\
\rho V_z^2 + P \\
\rho V_z r V_\theta \\
\rho V_r V_\theta \\
\rho V_r h_o
\end{bmatrix}, \quad F = \begin{bmatrix}
\rho V_\theta \\
\rho V_\theta V_z \\
r (\rho V_\theta^2 + P) \\
\rho V_\theta V_r \\
\rho V_\theta h_o
\end{bmatrix}, \quad G = \begin{bmatrix}
\rho V_r \\
\rho V_r V_z \\
\rho V_r r V_\theta \\
\rho V_r^2 + P \\
\rho V_r h_o
\end{bmatrix}
\]
and
\[
S_i = \begin{bmatrix}
0 \\
0 \\
0 \\
(\rho V_{\theta}^2 + P)/r \\
0
\end{bmatrix}, \quad S_{\text{vis}} = \begin{bmatrix}
0 \\
f_z \\
f_r \\
f \end{bmatrix} \tag{3.2}
\]

where \( P, \rho, V_z, V_\theta, V_r, e \) and \( h_o \) denote the static pressure, density, absolute velocity components, total energy and total enthalpy respectively and \( \hat{n} \) are the respective unit normal vectors in the polar cylindrical coordinates. The matrix, \( S_{\text{vis}} \) consists of the approximated viscous body force terms and the terms \( H_{\text{ro}} \) represent the effect of blade rotation.

The system of equations is completed with two additional relations to calculate pressure and total enthalpy which, for an ideal gas with constant ratio of specific heat capacities, \( \gamma (= c_p/c_v) \), are given as follows,

\[
P = (\gamma - 1) \left[ \rho e - \frac{1}{2} \rho (V_z^2 + V_\theta^2 + V_r^2) \right] \quad \& \quad h_o = e + \frac{P}{\rho} \tag{3.3}
\]

### 3.3 FINITE-VOLUME METHOD: CELL-VERTEX APPROACH

The governing equations are discretized using the finite-volume approach in a cell-vertex structured grid, first proposed by Ni (1981), where the flow variables are stored at the vertices (or the corners) of the cells. The control volume for each vertex (or node), \( \Delta Vol_{\hat{i}k} \), is defined by joining the centroids of the eight surrounding cells and calculated by averaging the volumes of its neighbouring cells, giving,

\[
\Delta Vol_{\hat{i}k} = \frac{1}{8} \sum_{n=1}^{8} \Delta Vol_n \tag{3.4}
\]
where the suffices, $i, j$ and $k$ are the indices of a node, $n$ refers to the surrounding cells and $\Delta Vol_n$ is the volume of the cell.

Figure 3.1: Cell-Vertex Scheme

Equation (3.1) applied to any control volume then becomes,

$$\Delta Vol_{ijk} \frac{dH_{ijk}}{dt} = -Q_{ijk}$$

(3.5)

where $Q$ is the flux change which will be defined in the next page.

The timed rate of change at any node $(i,j,k)$ is then given by,

$$\frac{dH_{ijk}}{dt} = \frac{Q_{ijk}}{\Delta Vol_{ijk}}$$

(3.6)

which is evaluated by taking a weighted average of the rate of change over the neighbouring cells (see figure 3.1) as suggested by Hall (1986), to give,

$$\frac{dH_{ijk}}{dt} = \frac{\sum_{n=1}^{8} \Delta Vol_{n}}{\sum_{n=1}^{8} \Delta Vol_{n}} \frac{1}{8} \sum_{n=1}^{8} \Delta Vol_{n}$$

(3.7)
where $Q_n$ consists of the net flux out of the cell, $n$, and the effects of the source terms given on the r.h.s. of equation (3.1), that is,

$$Q_n = \left( \sum FLUX \right)_n - (SOURCE)_n$$  \hspace{1cm} (3.8)

The value of the conserved variable at a particular cell face is obtained by averaging its values at the four corners and the total flux change across the cell is computed by taking the summation of the fluxes across the six faces of the cell (or volume) which is given by

$$\left( \sum FLUX \right)_n = \sum_{m=1}^{6} \left[ EdA_z + (F - Hr\omega) dA_\theta + GdA_r \right]_m$$  \hspace{1cm} (3.9)

where $dA_z$, $dA_\theta$ and $dA_r$ are the projected areas of these faces in the coordinate directions.

The term, $SOURCE$ consists of the inviscid and viscous effects given by $S_i$ and $S_{vis}$ respectively in equation (3.1). The inviscid term occurs only in the radial momentum equation to account for the centrifugal effect, which for cell, $n$, is given as

$$\left( SOURCE_i \right)_n = \left( \frac{\rho V_\theta^2 + P}{r} \right)_n \Delta Vol_n$$  \hspace{1cm} (3.10)

The viscous source terms are included in each of the three momentum equations and represent the viscous body forces which are approximated using a simple model whose details are given later in section 3.7.
3.4 ARTIFICIAL VISCOSITY

The finite-volume discretization amounts to central differencing and requires the addition of artificial viscosity or dissipation terms for stability. The dissipation terms are introduced explicitly to equation (3.6) to give,

\[
\frac{dH_{i,j,k}}{dt} = -\frac{Q_{i,j,k}}{\Delta Vol_{i,j,k}} + \frac{D_{i,j,k}}{\Delta Vol_{i,j,k}}
\]

(3.11)

where the dissipative terms are,

\[
D_{i,j,k} = d_{i,j,k} - d_{i-1,j,k} + d_{i,j+1,k} - d_{i,j,k+1} + d_{i,j,k+1} - d_{i,j,k+1}
\]

(3.12)

which consists of contributions from the r, \( \theta \) and z directions.

The terms, \( d \), in the equations are evaluated based on a blend of second and fourth order differences adopted from Jameson et al. (1981). The fourth order differences are added as background dissipation and the second order differences are added primarily to give oscillation-free shock resolution.

The second and fourth order dissipations have the following functional forms,

\[
d_{i+\frac{1}{2},j,k} = \alpha_{i+\frac{1}{2},j,k} \left[ \varepsilon_{i+\frac{1}{2},j,k}^{(2)} (H_{i+1,j,k} - H_{i,j,k}) - \varepsilon_{i+\frac{1}{2},j,k}^{(4)} (H_{i+2,j,k} - 3H_{i+1,j,k} + 3H_{i,j,k} - H_{i-1,j,k}) \right]
\]

\[
\varepsilon_{i+\frac{1}{2},j,k}^{(2)} = k^{(2)} \max(v_{i+1,j,k}, v_{i,j,k})
\]

\[
\varepsilon_{i+\frac{1}{2},j,k}^{(4)} = \max(0, k^{(4)} - \varepsilon_{i+\frac{1}{2},j,k}^{(2)})
\]

(3.13)

where \( v \) is the shock sensor parameter evaluated using undivided central difference of the static pressure,
and $A$ in equation (3.13) are user-specified constants which control the amount of artificial dissipation added. The term, $\alpha$, in the equation is the averaged ratio of the time-step and the control volume between two adjacent cells, which is,

$$
\alpha_{i,j,k} = \frac{1}{2} \left[ \frac{\Delta Vol_{i,j,k}}{\Delta t_{i,j,k}} \right] + \frac{\Delta Vol_{i+1,j,k}}{\Delta t_{i+1,j,k}} \right] \tag{3.15}
$$

where $\Delta t$ denotes the local time step derived by satisfying the CFL condition which is given in the next section.

### 3.5 NUMERICAL SCHEME: MULTI-STAGE RUNGE-KUTTA METHOD

The classical fourth order Runge-Kutta (RK) scheme is used to integrate equation (3.11) and the system is solved in four explicit stages of the form,

$$
\begin{align*}
H_{i,j,k}^{(0)} &= H_{i,j,k}^n \\
H_{i,j,k}^{(1)} &= H_{i,j,k}^{(0)} - \frac{\Delta t_{i,j,k}}{2\Delta Vol_{i,j,k}} \left( Q_{i,j,k}^{(0)} - D_{i,j,k}^{(0)} \right) \\
H_{i,j,k}^{(2)} &= H_{i,j,k}^{(0)} - \frac{\Delta t_{i,j,k}}{2\Delta Vol_{i,j,k}} \left( Q_{i,j,k}^{(1)} - D_{i,j,k}^{(1)} \right) \\
H_{i,j,k}^{(3)} &= H_{i,j,k}^{(0)} - \frac{\Delta t_{i,j,k}}{6\Delta Vol_{i,j,k}} \left( Q_{i,j,k}^{(2)} - D_{i,j,k}^{(2)} \right) \\
H_{i,j,k}^{(4)} &= H_{i,j,k}^{(0)} - \frac{\Delta t_{i,j,k}}{6\Delta Vol_{i,j,k}} \left( Q_{i,j,k}^{(0)} + 2Q_{i,j,k}^{(1)} + 2Q_{i,j,k}^{(2)} + Q_{i,j,k}^{(3)} \right) + \frac{\Delta t_{i,j,k}}{2\Delta Vol_{i,j,k}} \left( Q_{i,j,k}^{(2)} + 2Q_{i,j,k}^{(1)} + Q_{i,j,k}^{(3)} \right) + \frac{\Delta t_{i,j,k}}{2\Delta Vol_{i,j,k}} D_{i,j,k}^{(2)} \\
H_{i,j,k}^{n+1} &= H_{i,j,k}^{(4)}
\end{align*}
$$

where the dissipative terms are updated at each stage of the integration scheme.
The time-stepping method is adapted for the use of a constant Courant number for acceleration of convergence, which results in the time step being spatially-varying. The use of such time steps, known as local time-stepping, ensures stability and enhances the rate of convergence.

The local time step is evaluated for each cell and the nodal time step is taken to be the average of the time steps in the eight surrounding cells. The local time step for each cell is limited by the Courant, Friedrichs, Levy (CFL) condition on stability which requires the size of the step to be,

\[ \Delta t \leq \frac{CFL}{\left[A\right]_{\text{max}}} \]  \hspace{1cm} (3.17)

where \(|A|_{\text{max}}\) represents the spectral radii (i.e. the maximum eigenvalue) of the Jacobian matrices in the \((r, \theta, z)\) coordinates.

One reasonable estimate of the spectral radii is,

\[ \left[A\right]_{\text{max}} = \left(\frac{\Delta Vol}{\left(a + \left|V_N\right|\right)_{j} A_j + \left(a + \left|V_N\right|\right)_{k} A_k + \left(a + \left|V_N\right|\right)_{l} A_l}\right)^{-1} \]  \hspace{1cm} (3.18)

where \(a\) is the local speed of sound, \(A\) is the surface area of the face and \(V_N\) is the normal velocity through the face. The denominator term inside the brackets in equation (3.18) is taken to be sum of the average of \((a + \left|V_N\right|)\) multiplied by the face surface area in each of the three coordinate directions.

The CFL number, \(CFL\), given in equation (3.17) depends on the time-integration scheme; in this case of the classical four stage RK scheme, the constant is approximately 2.8 (Lobo, 1997).
3.6 GRID-SEQUENCING

Restricted by the CFL condition, the convergence to steady state for a hyperbolic system is generally slow. In the current solver, the technique of grid sequencing is used to accelerate the convergence rate. The idea is to begin the calculations with a very coarse mesh where larger time steps can be used to expel the transients quickly to the boundaries, and then complete the calculation by transferring the converged coarse solution using bilinear interpolation to a mesh refined by doubling the number of points in each direction.

A sequence of up to three meshes is implemented with the developed solvers. An overall efficiency of the factor 3.0 in reduction of the computational time was typically achieved with the technique (see Appendix I).

3.7 VISCOUS MODELLING

The use of the Euler solver has the main limitation of neglecting boundary layer blockage in predicting the real flow. Although not significant in most turbine flows, viscous effects can be significant in compressor blade flow. In such cases, an intermediate type of method can be employed in which the inviscid solution is coupled, in a single calculation, with a very simple approximation to model viscous effects.

One technique that has been reported to give very convincing results is that suggested by Denton (1986, 1990). Denton's methods are based on a 'distributed loss' model in which the viscous effect is approximated by obtaining the flow shear stresses along the solid surfaces. The approximated shear stresses are then distributed through the flow to provide components of viscous forces on each control cell.

Denton (1986) first proposed the possibility of simple modelling where the surface shear stresses are directly computed from the classical skin friction coefficient and the relative dynamic head of the surface streamline, i.e. \( \tau_w = \frac{1}{2} \rho W^2 C_f \). In this work, the skin friction, \( C_f \) is given as a constant (determined empirically) and an algebraic
function (a power law) is used to distribute the shear stress away from the solid boundaries. Despite its simplicity, the approximation has led to realistic results. Denton (1990) then reported an improved extension where the dissipative forces are computed using the log law with the mixing length model. Both his methods are implemented here to model viscous losses.

It is acknowledged that the modelling of viscous effects in the current solver codes is based entirely on the method proposed by Denton (1986, 1990) where full details can be found. The more recent method of Denton (1990) is briefly outlined here.

In this implementation, the viscous forces are obtained from a thin layer approximation where only the shear stresses on the streamwise and bladewise faces of the control volume are considered. The shear stress on the boundary walls is computed assuming that the first grid away from the solid surfaces lies either in the laminar or the logarithmic region of a turbulent boundary layer.

If the flow is laminar, the wall shear stresses are determined using the laminar frictional law, which gives the value of $C_f$ to be,

$$C_f = \frac{2}{Re} \quad (3.19)$$

where $Re$ is the local Reynolds number based on the distance from the blade surfaces or end walls as the characteristic length, and the local relative velocity at the grid point as the reference velocity.

In the latter case, the expression for the log law boundary layer profile is used. In particular, the empirical equation derived by Millikan is implemented here. Millikan's empirical expression is,

$$W^\circ = 2.5 \ln Y^\circ + 5.2 \quad (3.20)$$
And since,

$$W^\theta = \frac{W}{W^\theta} = \sqrt{\frac{\rho W^2}{\tau_w}} = \left(\frac{2}{C_f}\right)^{\frac{1}{2}} \text{ and } Y^\theta = \frac{W^\theta Y}{\nu} = \sqrt{\frac{C_f}{2}} \text{Re} \quad (3.21)$$

Equation (3.21) can be rewritten as,

$$\left(\frac{2}{C_f}\right)^{\frac{1}{2}} = 2.5 \ln \left(\left(\frac{C_f}{2}\right)^{\frac{1}{2}} \text{Re}\right) + 5.2 \quad (3.22)$$

Using equation (3.22) the skin friction can be approximated. Denton gives the approximation for $C_f$ as,

$$C_f = -0.001767 + \frac{0.03177}{\ln(\text{Re})} + \frac{0.25614}{(\ln(\text{Re}))^2} \quad (3.23)$$

For the elements away from the boundary walls, the Von-Karman equation is used to predict the mixing-length eddy-viscosity and subsequently to calculate the shear stresses. Viscous forces acting on each volume are found by summing the shear stresses on the streamwise and bladewise faces of the cell whose components are included as the source term, $S_{vis}$, in the r.h.s. of each of the three momentum equations of equation (3.1).

It is noted that the idea to include the viscous effect through this approximation is a simple one which does not aim to achieve the same level of detail of a viscous flow as predicted by the Navier-Stokes equations. Instead, the technique is employed to determine only qualitative effects of the viscous flow and the compromise is made in exchange for speed in the computation.
3.8 GRIDDING AND BOUNDARY CONDITIONS

The development of the Euler solver code is intended to give the solution of the flow past a cascade blade row arrangement. A typical cascade computational domain set up with simple sheared H-topology is shown below.

![H-Grid Topology Diagram]

**Figure 3.2: Turbomachinery Cascade Represented in H-Grid Topology**

H-, O- and C- type grids are examples of commonly used grid topologies found in numerical flow calculation for turbomachinery cascades. It is known that no one type of grid is ideal for all types of blade row (Dawes and Denton, 1998), and in three-dimensional modelling, mesh generation can be complicated when grids at different spanwise grid surfaces need to be linked together. As a result, some popular codes (for example, Denton, 1982, Dawes, 1987 and He, 1993) retain a simple sheared H-grid for three-dimensional single blade and multistage calculations of turbomachinery blade flows.
In the current application, the mesh generation procedure must be efficient since it is required following every blade modification. Therefore, to alleviate the overall computational time, simple sheared H-gridding is applied here.

In the cascade calculation, four different types of boundaries are required: inlet, outlet, solid walls and periodicity.

At the inlet, the total pressure and temperature profiles and two absolute flow directions in the streamwise and meridional plane are specified. The static pressure is extrapolated from the interior flow field domain, which is used in conjunction with the isentropic relation to calculate the density and velocity.

At the downstream boundary, the exit static pressure is held fixed on the hub and the radial variation is obtained via the radial equilibrium conditions. Alternatively, if the radial exit pressure distribution is known, the specification may be applied directly. The remaining flow variables are then extrapolated from those at the interior nodes.

At the solid surfaces of the blade including the shroud and hub walls, normal convective fluxes through the walls are forced to zero, and the surface flow satisfies the tangency condition and slips along the blade surfaces.

A periodic or a cyclic condition is imposed to model the symmetry of flow between each sector of the cascade. This condition is applied upstream of the leading-edge and downstream of the trailing-edge where the values at two corresponding points are set to be equal.
3.9 CONVERGENCE CRITERIA

For all the test cases considered here, the iterative analysis procedure is completed when the solutions satisfy the following convergence criteria,

1. Mass flow rate at each stream-wise grid line agrees with the inlet mass flow to within 0.5 percent.

2. The maximum change in the absolute velocity anywhere in the flow field divided by the root-mean-square of the absolute velocities in the flow field is smaller than $1.5 \times 10^{-5}$.

3. Inlet mass flow does not change by more than 0.1 percent over 50 time steps.

The above settings were decided based on several numerical trials indicating that the pseudo-steady solutions were obtained when these criteria are satisfied. In essence, the criteria require that the timed rate of change (i.e. the r.h.s. of equation 3.11) has become sufficiently small between iterations indicating that the flow field has stabilized to a steady state.

In chapter 4, verifications of the two- and three-dimensional solver codes are presented. All the chosen test cases are practical examples of turbomachinery flow where experimental data are available for comparison with the computation. The real advantage of the time-marching method is in the calculation of three-dimensional shocked flow, therefore the accuracy of the 3-D method is verified with transonic cases.
CHAPTER 4
VALIDATION OF THE EULER FLOW SOLVER

4.1 INTRODUCTION

This chapter presents the validation of the flow solver whose development is described in the previous chapter. Since the solver codes are integral to the inverse procedures and their solutions are used directly to determine the blade shape, it is important to first establish the reliability of their steady viscous solutions.

Several test cases, both two- and three-dimensional, are used for the validations. All the test cases presented are practical examples of actual turbomachinery flow and have well-documented experimental data with which the flow results can be compared and assessed.

In the validation of the two-dimensional analysis, the mid-span section of the first vane and first rotor blade of a low-speed, axial flow turbine stage tested by Dring et al. (1982) are used as the test subjects. Here, the flows through the stator and the rotor cascades are modelled separately and the surface flow data are compared with the experiments.

For the three-dimensional analysis, the test cases presented are two transonic subjects taken from AGARD Report 275 (1985) entitled: Test Cases for Computation of Internal Flows in Aero Engine Components which are catalogued specifically for the verification of computational results.

The chosen test cases are an annular turbine nozzle blade and a NASA designed fan, rotor 67. Both are highly three-dimensional and transonic, and would therefore stringently test the capability of the solver code in resolving complicated viscous transonic turbomachinery flow.
4.2 UTRC STATOR AND ROTOR CASCADE

The United Technologies Research Center (UTRC) turbine model has three rows of blades; first vane (stator), first blade (rotor) and second vane (stator). In the experiment, only the first vane and the first rotor were installed and tested. Details of the experiments can be referred to in Dring et al. (1981, 1982).

The basic geometrical features of the blades and their nominal operating conditions are summarized in table 4.1.

<table>
<thead>
<tr>
<th></th>
<th>Stator</th>
<th>Rotor</th>
</tr>
</thead>
<tbody>
<tr>
<td>No. of Blades</td>
<td>22</td>
<td>28</td>
</tr>
<tr>
<td>Axial Chord (m)</td>
<td>0.1506</td>
<td>0.1610</td>
</tr>
<tr>
<td>Span (m)</td>
<td>0.1524</td>
<td>0.1524</td>
</tr>
<tr>
<td>Absolute Inlet Flow</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Angle (deg)</td>
<td>0.00</td>
<td>67.50</td>
</tr>
<tr>
<td>Stagger Angle (deg)</td>
<td>49.5</td>
<td>32.7</td>
</tr>
<tr>
<td>Aspect Ratio (Span/Axial Chord)</td>
<td>1.01</td>
<td>0.95</td>
</tr>
<tr>
<td>Reynolds Number, Re</td>
<td>$5.9 \times 10^5$</td>
<td>$5.5 \times 10^5$</td>
</tr>
<tr>
<td>Rotational speed, rpm</td>
<td>Not Applicable</td>
<td>410</td>
</tr>
</tbody>
</table>

Table 4.1: Basic Specifications of the UTRC Stator and Rotor

4.2.1 Description of the Experiment and Experimental Data

In the experiment, the aerofoils are scaled to five times the engine size. The large-scale rig allowed the use of extensive instrumentation and gave a Reynolds number typical of the value in high-pressure turbines while being operated at the nominal model running
condition of 410 rpm. Available experimental data included both steady and unsteady
data, the former being used here for verification of the computational results.

At various span locations, the surface static pressure was measured by pressure taps
positioned around the perimeters of the blade sections. These data was presented in the
form of a pressure coefficient ($C_p = \frac{P_0^{STA1} - P}{\frac{1}{2} \rho v^2_{STA2}}$) based on the total pressure at the
inlet (station 1) and the dynamic pressure at the exit of the first vane (station 2).

A schematic diagram of the test rig showing the locations of station 1 (STA 1) and
station 2 (STA 2) is given in figure 4.1.

![Figure 4.1: United Technologies Research Center Large Scale Rotating Rig
(Schematic Diagram Reproduced from Dring et al., 1981 Report)](image)

### 4.2.2 Numerical Analysis

In both cascades, the high aspect ratio (see table 4.1) of the blades isolated the end-wall
effects and gave rise to a smooth and nearly two-dimensional flow near the mid-span
(see Dring et al., 1982).

In addition, it is also noted that the steady state flow over the rotor had only a weak
dependence on the axial gap from the adjacent row of vanes upstream. Based on these
observations, two-dimensional simulations of the flows past the mid-span sections of both blades are performed separately for the validations. In both computations, viscous modelling is included. The viscous shear stresses are approximated with a constant wall friction factor, \( C_f \) and distributed with an algebraic power law constant, \( n \). The chosen constants, \( C_f = 0.01 \) and \( n = 10 \) are typical values suggested by Denton (1986) which are also confirmed by numerical experimentation here. The UTRC experiments were carried out at three flow coefficients \( \frac{V_{c}}{r \omega} = 0.68, \frac{V_{c}}{r \omega} = 0.78 \) and \( \frac{V_{c}}{r \omega} = 0.96 \). In the current analysis, the flow condition corresponding to \( \frac{V_{c}}{r \omega} = 0.78 \) is used.

### 4.2.2.1 Grid Definition

Simple sheared H-grid topology is employed to define the computational domains for the cascades. The blunt leading-edges of both cascades are modelled correctly but their blunt trailing-edges are replaced by short wedges simulating the recirculating flow zone downstream of the edges. This is a common practice to ensure that the flow leaves the blade smoothly and to eradicate non-physical prediction of the steady state of the unsteady vortex shedding. This was described recently by Dawes and Denton (1998).

The grid definitions for both cascades are similar and each employs 1911 grid points; 21 points in the blade-to-blade direction, 91 points in the stream-wise direction with 51 points from the blade leading-edge to its trailing-edge for the stator and 71 points for the rotor. Non-uniform pitch-wise grid distribution is used in which more grids are concentrated near the blade surfaces and the finest cells adjacent to the blade walls are about 1/6 that at mid-pitch.

Figure 4.2 shows the generated grids; on the left is the computational domain generated for the stator and on the right that for the rotor.
The experimental flow conditions measured upstream and downstream of the two airfoils are implemented as the inlet and outlet boundary conditions in the computations. At the upstream boundary, the inlet total pressure, temperature and absolute inlet angle are imposed and assumed to be uniform in the tangential direction. At the exit, the static pressure is fixed.

In the flow computation of the rotor section, a constant rotating tangential velocity of 29.44 m/s is specified to simulate the nominal operating rotational speed of 410 rpm.

4.2.2.2 Comparisons with Experimental Data

The comparisons between the experimental and computed pressure coefficient, $C_p$, along the pressure and suction sides of the stator and rotor cascades are shown in figure 4.3 and 4.4 respectively.

Both figures show good correlation between the numerical and experimental results except for some discrepancies near the leading- and trailing-edges. The differences in the results near the leading-edges arise as a result of modelling the leading-edge with
the simple sheared H-type grid. Employing O or C-type grid topology would lead to better leading-edge resolution. However, for the reasons given in Chapter 3, H-grid is applied here.

Despite the inaccuracies near the leading-edges in these cases, the errors incurred at the front do not convect downstream nor adversely affect the solution substantially; from the figures, the solutions (blue lines) can be observed to match the experimental distributions (circles) satisfactorily.

![Graph](image)

**Figure 4.3**: Validation: UTRC Stator
Comparison of Numerical and Experimental Pressure Coefficient, $C_p$

![Graph](image)

**Figure 4.4**: Validation: UTRC Rotor
Comparison of Numerical and Experimental Pressure Coefficient, $C_p$
4.3 MEL ANNULAR TURBINE NOZZLE

The first three-dimensional test case is a low hub-to-tip ratio annular turbine nozzle guide vane. As was mentioned at the start of this chapter, it is taken from AGARD Report 275, which is meant to serve as the basis for validation of computational codes. As such, the test case (a sixth-scale prototype) has well-defined experimental boundary conditions that can be applied directly in the flow simulation, and is chosen because it has the characteristics of a real working transonic machine. In particular, it has all the features of a last stage L.P. steam turbine nozzle; namely, low hub-tip radius ratio, high turning, high casing flare, highly turbulent flow (see table 4.2) which is transonic at the hub and subsonic at the casing. Furthermore, this nozzle has the added complications of leaning leading- and trailing-edges in the meridional plane and a non-uniform profile cross-section.

<table>
<thead>
<tr>
<th>ANNULAR NOZZLE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Blade Pitch (Degree)</td>
</tr>
<tr>
<td>Hub to Tip Ratio</td>
</tr>
<tr>
<td>Casing flare (Degree)</td>
</tr>
<tr>
<td>Reynolds Number, Re</td>
</tr>
<tr>
<td>Turning Angle (Degree)</td>
</tr>
<tr>
<td>Hub</td>
</tr>
<tr>
<td>Tip</td>
</tr>
</tbody>
</table>

Table 4.2: Essential Features of the Annular Nozzle Blade

With so many features that are difficult to predict, this blade is a suitable, fully three-dimensional, test case for validating the numerical solution.
4.3.1 Description of the Experiment and Experimental Data

The annular turbine test section was tested in the MEL (Marchwood Engineering Laboratories) supersonic cascade rig. The test rig was continuously operating and had a closed loop design with air as the working fluid. Comprehensive details of the rig, facility and instruments are given by Ball et al. (1985) in section VIII.2 of AGARD report 275 and Spurr (1980a, 1980b).

Figure 4.5 shows the working section of the annular cascade with details of the inlet and outlet sections.

![Diagram of Working Section of Annular Cascade](image)

**Figure 4.5: Working Section of the Annular Cascade**  
(Schematic Diagram Reproduced from AGARD Report 275)

The casing geometry has been carefully designed to ensure smooth entry flow and is seen here to have a flared opening of about 55 degrees.
In the experiment, both static and stagnation flow-field pressures were recorded. The measurements were made by pressure probes along several transverse planes and by pressure taps positioned around the perimeter of the blade at three blade heights, namely, near the blade root, at mid-span and near the tip. The surface static pressure distributions measured using the taps are the given data for comparison with full 3-D inviscid or viscous calculations (Ball et al., 1985). These data were first used as such by Spurr (1980a, 1980b) in his numerical computation in inviscid flow. The same experimental data are used here to verify the viscous results of the current computation.

4.3.2 Numerical Analysis

In this analysis, the viscous shear stresses are approximated using the mixing length model, and are calculated on both the blade surfaces and on the end-walls.

The experimental total pressure and temperature are held fixed at the inlet as the inflow boundary conditions, and the static pressure distribution measured at the transverse position along the perforated plate (see figure 4.5) is applied as the numerical outlet condition. The pressure ratios (outlet static/inlet total) at the hub and tip are 0.281 and 0.608 respectively.

4.3.2.1 Grid Definition

Figure 4.6 shows the blade assembly in the meridional plane. The computational domain is set up to represent the exact geometry of the test section. It is defined with H-grid topology and employs a total of 101761 nodes; 29 points in the blade-to-blade direction, 121 points in the stream-wise direction with 63 points from the blade leading-edge to its trailing-edge and 29 points in the span-wise direction from hub-to-shroud.
The mesh is extended at about one axial chord upstream of the leading-edges, and the downstream boundary is set up to coincide with the location where the perforated plate was positioned in the experiment (so that the measured outlet condition can be applied directly); this is approximately 0.75 times the tip axial chord downstream at the tip and 1.5 times the hub axial chord downstream at the hub. The generated mesh is shown in figure 4.7.
Figure 4.7: Computational Mesh for the Annular Nozzle
4.3.2.2 Comparisons with Experimental Data

The experimental surface static pressure distributions measured at the three blade heights are compared with the computational results. The pressure tapping locations do not coincide exactly with the surface computational nodes at any single span-wise grid position. As a result, linear interpolations are carried out to post-process the data so that direct comparisons can be made.

The calculated surface static pressures are normalized with the upstream stagnation pressure and are plotted against the percentage axial chord from the blade leading-edge. Comparisons of the measured and calculated data at the three blade height positions are shown in figures 4.8, 4.9 and 4.10.

In all three figures, only small isolated discrepancies are observed and the numerical solutions are, on the whole, in close agreement with the measured values. The viscous solution therefore agrees favourably with the high Reynolds number experiment.

The flow field results are described in more detail in the next section.
Along Pressure Tappings: Near Hub

Figure 4.8: Surface Static Pressure Distribution near Hub Pressure Tapping Location

Along Pressure Tappings: Near Mid-Span

Figure 4.9: Surface Static Pressure Distribution near Mid-Span Pressure Tapping Location

Along Pressure Tappings: Near Casing

Figure 4.10: Surface Static Pressure Distribution near Casing Pressure Tapping Location
4.3.3 Overview of the Flow Field

The meridional geometry of the nozzle represents a convergent-divergent channel; the divergence aft of the section is considerable and leads to almost 50% increase in the annulus area between the inlet and exit of the cascade. The increase in radius along the casing and streamtube divergence tends to turn the swirling flow outwards and off-loads the nozzle. This effect is predicted accurately by the numerical computation as evident from the surface pressure distributions in figures 4.8, 4.9 and 4.10.

The highest load is experienced at the hub where the flow has a maximum Mach number of about 1.8. The flow remains transonic over the lower half of the blade before becoming completely subsonic near the casing where the load is substantially less.

Mach number contours in the blade-to-blade plane near the hub, mid-span and casing are shown in figure 4.11.

Figure 4.11: Computed Mach Number Contours at 4%, 50% & 96% Blade Height
(Contour Interval: 0.05)
The flow is accelerated as it progresses through the narrowing cross-sectional area in the blade-to-blade passage and the turning occurs mainly at the rear portion of the blade. The net result is that the loading is concentrated at the aft of the blade, the effect of which is also clearly seen in the surface pressure distributions shown in figures 4.8, 4.9 and 4.10.

From the literature of the test rig (Spurr, 1980b), the nozzle section has undergone intensive designs. Design was primarily concerned with the suitable contouring of the inlet section (as shown in figure 4.5) to accommodate the high casing flare of the cascade. The aim was to minimize the deceleration of the flow in the diffusing section upstream of the cascade so that the boundary layer does not separate. This problem was satisfactorily eradicated through careful adjustment of the curvature of the wall and checking for flow separation in the experiment. In the experiment, flow separation was a major concern because it can complicate measurements of the flow field. In practical applications of turbomachinery, flow separation has more serious implications; first and foremost it involves losses and has detrimental effects on the overall behaviour of the machine.

Boundary layer separation can also occur due to poor design of the blade shape. In turbine flow, although the boundary layer is usually thin due to the favourable pressure gradient of the accelerating flow, diffusion following peak suction acceleration can be strong enough to trigger separation of the surface boundary layer. The condition is further aggravated by shock interaction if the flow becomes locally supersonic.

The prediction of separation in a three-dimensional flow is a difficult task, but a rough guide from compressor and diffuser tests to avoid high losses due to boundary layer build up is that the diffusion factor, \( D_f \), must be less than 0.6 where,

\[
D_f = \frac{V_{\text{max}} - V_{\text{exit}}}{V_{\text{inlet}}} \tag{4.1}
\]

The criterion is based on empirical results of cascade tests (see Appendix II) where it is observed that losses increase rapidly when the limit is reached, due to the thickening of the boundary layer on the suction side of the blade.
In this nozzle, the blade loading is highest at the hub, therefore the velocity and its associated diffusion over the suction surface is strongest here. Figure 4.12 shows the surface velocity marked with the line representing the limiting velocity level corresponding to $D_F = 0.6$ (red-dashed line).

From the figure, the diffusion aft of the blade exceeds the limit placed by the 2-D empirical criterion. Although there is no separation in the current flow, the diffusion aft of the blade is seemingly high and reducing it would be a worthwhile objective both from a practical point of view and for study*. This is carried out using the present blade design method (section 6.2.2, chapter 6).

* The diffusion rate may also be quantified using the exit flow velocity, i.e. $D_F = \frac{V_{max} - V_{exit}}{V_{exit}}$, which is 0.209 in the current case.
4.4 NASA LEWIS ROTOR 67

The second three-dimensional test case is a transonic fan blade and is also an AGARD test case (AR 275, 1985). The rotor considered here, rotor 67, is the first stage rotor of a NASA Lewis designed, two-stage fan (Urasek et al., 1979). The basic design specifications are shown in table 4.3.

<table>
<thead>
<tr>
<th>Rotor 67</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>No. of Rotor Blades</td>
<td>22</td>
</tr>
<tr>
<td>Rotational Speed (rpm)</td>
<td>16043.0</td>
</tr>
<tr>
<td>Mass Flow (kg/s)</td>
<td>33.25</td>
</tr>
<tr>
<td>Pressure Ratio</td>
<td>1.63</td>
</tr>
<tr>
<td>Rotor Tip Speed (m/s)</td>
<td>429</td>
</tr>
<tr>
<td>Tip Clearance at Design Speed (cm)</td>
<td>0.061</td>
</tr>
<tr>
<td>Inlet Relative Mach No.</td>
<td>1.38</td>
</tr>
<tr>
<td>Rotor Aspect Ratio</td>
<td>1.56</td>
</tr>
<tr>
<td>Based on Averaged span/Root Axial Chord</td>
<td></td>
</tr>
<tr>
<td>Rotor Aspect</td>
<td>Hub 3.17</td>
</tr>
<tr>
<td></td>
<td>Tip 1.29</td>
</tr>
<tr>
<td>Tip Diameter (m)</td>
<td>Inlet 51.4</td>
</tr>
<tr>
<td></td>
<td>Outlet 48.5</td>
</tr>
<tr>
<td>Hub/Tip Radius Ratio</td>
<td>Inlet 0.375</td>
</tr>
<tr>
<td></td>
<td>Outlet 0.478</td>
</tr>
</tbody>
</table>

Table 4.3: Basic Specifications of the NASA Rotor 67

4.4.1 Description of the Experiment and Experimental Data

Extensive experiments were performed by Strazisar et al. (1989) on rotor 67 using aerodynamic probes and a laser anemometer. The experimental measurements were compiled by Wood et al. (1985) in section VI.2 of the AGARD report 275. Details of the laser anemometer (LA) system and probe instruments can be found in Strazisar (1985), and Powell et al. (1981).
Figure 4.13 shows the meridional view of the rotor blade with the experimental LA and aerodynamics survey points defined with respect to axial (AP) and radial (RP) coordinates.

Radial measurements of the total static pressure, total temperature and swirl flow angle were made at two designated survey locations located upstream (at AP = -4.632 cm) and downstream (at AP = 8.852 cm) of the rotor. These locations are denoted as Aero Station 1 and Aero Station 2 respectively (see figure).

Measurements were also made in the pitch-wise direction from which blade-to-blade Mach number contour plots were constructed.

These data and plots are used to verify the computations here.
4.4.2 Numerical Analysis

As in the first 3-D test case (section 4.3), the numerical analysis of the flow past the cascade is performed using the developed three-dimensional solver with viscous effects included using the mixing length model.

The numerical boundary conditions are applied using the experimental data. At the upstream boundary, the presence of the boundary layers on the hub and casing wall is accounted for by the total pressure and temperature profiles whose distributions are obtained from the experiment. In addition, the absolute inlet swirl angle and meridional pitch inlet angle are also specified and are fixed at each span-wise grid and assumed to be uniform in the tangential direction.

At the outlet, the static pressure at the hub is prescribed and the pressure gradient from the hub to tip is then defined by the radial equilibrium condition.

4.4.2.1 Grid Definition

The complete set of geometrical data was given on fourteen blade surface sections including one at the hub and one at the casing of the machine. The data consisted of the axial and radial co-ordinates as well as the angular co-ordinates of the upper and lower blade surfaces. In this analysis, no tip leakage flow is modelled and the tip of the blade is assumed to coincide with the casing.

The inlet and outlet boundaries are generated by extending the computational domain fore and aft of the rotor blade respectively. The locations of the inlet and outlet boundary extensions used in this study are based on the proportions used by Pierzga and Wood (1985) in the accompanying computational study of the experiment using the three-dimensional solver code of Denton (1982). The upstream boundary is located 1.5 times the tip axial chord upstream of the tip (tip ratio) and 0.7 times the hub axial chord upstream of the hub (hub ratio). The downstream boundary is located two tip axial chords downstream at the tip and one hub axial chord downstream at the hub. Between the hub and tip, the upstream and downstream grid boundary limits are determined by
linearly interpolating the ratios used for the hub and tip and multiplying by the axial chord at the respective span position.

Simple sheared H-grid topology is used to generate the computational mesh in the blade-to-blade passage as well as from the hub to the casing. Non-uniform grid distribution is used in which more cells are concentrated near the blade surfaces and the end walls. Final solutions (see grid dependency test in Appendix III) presented were obtained with the computational grid, which employs 118581 grid points; 29 points in the blade-to-blade direction, 141 points in the stream-wise direction with 77 points from the blade leading-edge to its trailing-edge and 29 points in the span-wise direction from hub-to-shroud. A three-dimensional view of the generated grid for the rotor blade is shown in figure 4.14.

\[ \text{Figure 4.14: Three-Dimensional Grid of NASA Rotor 67} \]
4.4.2.2 Comparisons with Experimental Data

In the experiment, a wide range of data points, from maximum flow to near-stall conditions, were taken. The most detailed experimental studies were made near the peak efficiency and at the near-stall condition. Therefore, the flow results computed at these two flow conditions are compared with the available experimental data.

The comparisons of the computed and experimental performance maps in terms of the total-to-total adiabatic efficiency and total pressure ratio are shown in figure 4.15 where the open square legends denote the experimental values and the solid circles are the numerical points. The numerical performance maps are constructed on the basis of calculations performed at 13 different values of mass flow rate, varied by changing the exit static pressure fixed at the hub but at the same design speed. The computed mass flows are non-dimensionalized with respect to choked mass flow and are plotted against the corresponding efficiency and total pressure ratio.

The calculated choked mass flow is 34.4 kg/s which is 1.3% lower than the experimentally measured value of 34.96 kg/s. Underestimation of the choked mass flow rate in the region of 1 % is also reported in other viscous calculations, for example, Jennions and Turner (1993), Arnone (1994) and Arima et al. (1997). A difference of this magnitude is thought to be acceptable considering the possible numerical and experimental uncertainties.

The left-most point on the computed curves represents the minimum mass flow rate at which the computation will converge. As the pressure is raised above that specified for the left-most point, a region of reversed flow develops through the inlet and no solution is possible. The last point on the left is thus taken to be the numerical stall point on the curve.
Both performance maps coincide reasonably with the experiment. The predicted total pressure ratio across the operating range is well predicted with no obvious discrepancies throughout the working range. Greater flow range and slightly higher efficiency are however predicted. These overestimations are a direct consequence of neglecting the tip leakage flow since losses due to the dissipation and mixing of the leakage flow are not introduced. The current results tally with the study made by Jennions and Turner (1993) who investigated the effect of changing the clearance gap of rotor 67.

The computed solutions are also assessed by comparing with the experimental span-wise distribution of circumferential averaged thermodynamic quantities measured upstream and downstream of the rotor. Experimental and computed total pressures, total temperatures and flow angles at the rotor survey stations (i.e. Aero Stations 1 and 2, see figure 4.13) are shown in figure 4.16, 4.17 and 4.18 respectively.
Figure 4.16: Total Pressure Profiles at Aero Stations 1 & 2

Figure 4.17: Total Temperature Profiles at Aero Station 2

Figure 4.18: Swirl Angle Profiles at Aero Station 2
At both the upstream and downstream survey stations, the predicted radial profiles of the total pressure ratios generally agree well with the experiment at the near-peak efficiency and near-stall conditions. At Aero Station 1, the profiles predicted under both operating conditions are very similar to each other and both match closely with the experimental data; there is only a slight difference for values below 20% radial span from the hub. The pressure profile predicted at Aero station 1 is directly related to the total pressure specified far upstream as the inlet boundary conditions; the good correlation is therefore expected and the discrepancies near the hub are a result of approximation when the experimental inlet total pressure distribution is interpolated to coincide with the grid points.

At the exit (Aero Station2), there is also good agreement between the solutions and the experimental total pressure ratios except for the point at the casing (100% span). The discrepancy is largely due to the exclusion of the tip leakage flow in the current simulations; the blade unloading at the tip region associated with an appreciable drop in pressure due to leakage flow, is clearly absent. The imposed wall boundary condition also plays a part since the surface velocities are allowed to slip in the current computation instead of coming to rest (i.e. the no-slip condition) as they would in a real viscous condition; the pressure drop near the surface is therefore less pronounced. In addition, the simple viscous model would have contributed to the discrepancies since full viscous effects are not accounted for. The profile resolution near the hub wall could not be checked as no experimental data at the hub were given.

The correlations of the total temperature ratio and the absolute exit swirl angle profiles near peak efficiency and near stall conditions with the experimental data are also satisfactory. Figure 4.18 indicates some overestimation in the exit flow angle predictions, up to 2 and 5 degrees off between 20% and 80 % of the radial blade span location near peak efficiency and stall conditions respectively.
4.4.2.3 Comparisons of Blade-to-Blade Flow

The computed Mach number distributions along three blade heights and at several pitch positions are also checked. The Mach number distributions are plotted against the normalised axial distance from the blade leading-edge position. Zero and one hundred percent axial positions thus coincide with the blade leading- and trailing-edges respectively. Figure 4.19 shows the comparisons of the predicted and actual Mach number levels along quasi-streamlines at 20%, 50% and 80% of blade pitch for blade heights of 30% and 70% for the flow condition near peak efficiency. The experimental values are represented by the solid square and the computational data is represented by the blue solid lines.

From the plots, the numerical results are observed to be in good agreement with the experiment; the shock positions and strengths are in general well predicted, and the computed Mach number distributions match the test data closely. There are, however, greater discrepancies downstream of the blade where the Mach number level is underpredicted at 70% blade height. The trailing wake region of the transonic blade is highly turbulent and accurate flow prediction in the wake is known to be difficult. The same observations were also shown by Calvert et al. (1997) using the Navier-Stoke solver, TRANSCode and the Baldwin-Lomax turbulence model. In view of that, the discrepancies are likely to be a result of the limitations of the turbulence modelling. It is possible that more advanced turbulence models can be implemented to improve the result; but as pointed out by Denton (1990), in the current situation where even sophisticated turbulence models predict only qualitatively accurate results, there is little justification in using more complex models. While the current implementation of the turbulence (or viscous) model is perhaps in its simplest form, the results obtained here agree well with those calculated by the most commonly used Baldwin-Lomax model.
1) 30% blade Height
II) 70% blade Height

Figure 4.19: Stream-wise Distributions of Relative Mach Number at the Near-Peak Efficiency Condition
In addition to the surface Mach number data, blade-to-blade Mach number contours at 30%, 70% and 90% blade height at near-peak efficiency and near-stall conditions are also available for comparison with the numerical results. Figures 4.20 and 4.21 show the experimental Mach number contour plots near peak efficiency and at near-stall conditions respectively. The experimental plots shown are all reproduced from the article of Pierzga and Wood (1985).

In the laser velocimetry measurements of the flow velocity, there was a response lag during which the particles (or seeds) did not accelerate as quickly as the flow. It was believed that the beginning of the shock was correctly captured by the experiment, but the shock was somewhat smeared. Taking this into consideration, Pierzga and Wood (1985) estimated the locations of the experimental shock and these were indicated by the dashed lines in the experimental plots.

All the Mach number contour plots are made with the contour interval of 0.05 as used in the experimental plots. Under both operating conditions, good agreement in terms of the flow pattern and the Mach number values can be observed. Greater discrepancies are observed in the plots at 90% blade height in which the shock formation is markedly more defined in the numerical prediction. Since the section is near the casing, it was initially thought that the smeared experimental contours are due to the leakage flow and therefore the absence of its modelling in the current simulation would give rise to the observed differences. However, this hypothesis becomes less conclusive when the results of other computations modelling the tip flow (for example, Jennions and Turner 1993, Arnone, 1994 and He and Denton, 1994) show similar differences. The effect of tip leakage modelling was not studied in the current work but, based on the results of these authors, experimental shortcomings (i.e. the response lag in measurement) in measuring highly shocked flow are the more likely cause for the less-defined, more smeared shock patterns in the experimental contour plot.
I) 30% Blade Height

II) 70% Blade Height

III) 90% Blade Height

Figure 4.20: Experimental (left) and Computed (right) Contours of Relative Mach Number at the Near-Peak Efficiency Condition; Contours at Intervals of 0.05; Dashed Lines are Estimated Shock Positions from Pierzga and Wood (1985)
Figure 4.21: Experimental (left) and Computed (right) Contours of Relative Mach Number at the Near-Stall Condition; Contours at Intervals of 0.05; Dashed Lines are Estimated Shock Positions from Pierzga and Wood (1985)
4.4.3 Discussion of Results: Effect of Viscous Modelling

The issue of how the flow computational result is influenced by the inclusion of the inviscid/viscous modelling needs to be investigated. Both Euler (pure inviscid and with inviscid/viscous coupling) and Navier-Stokes computations are thus performed to investigate the relative performance of their solutions.

The present Euler code can perform flow simulation in the pure inviscid limit or with inviscid/viscous modelling included. The study is therefore carried out by comparing the result of the current computation (inviscid/viscous solution) with that when pure inviscid flow is assumed and with a third calculation solving the three-dimensional Navier-Stokes equations. All the flow analyses are carried out at the near-peak efficiency condition.

The three-dimensional Navier-Stokes solver used in the study is that of Dawes (1987). The code (BTOB3D) is based on finite-volume formulation and uses a time-marching scheme consisting of a two-step explicit and one-step implicit scheme derived as a pre-processed simplification of the Beam and Warming algorithm. Dawes' code is well validated and is used by several establishments in the simulation of subsonic and transonic flows in both axial and centrifugal configurations (Goto, 1992, Hathaway et al., 1993).

The Navier-Stokes solver also uses structured H-grid topology but employs the cell-centre implementation in which the flow variables are located at the centroids of the cell volumes. Surface flow results presented here are thus obtained by extrapolating the flow solutions of the interior cells adjacent to the blade walls. To be consistent with the previous computations, the simulation using Dawes' code is also carried out without the modelling of tip leakage flows.

Comparisons of the results are carried out with the predicted Mach number contours. These are made at several blade heights as shown in figure 4.22.
Figure 4.22: Comparisons of Mach Number Contours: Inviscid, Inviscid/Viscous and Viscous Solutions
As shown in the figures, the flow details correlated well, particularly the shock locations and overall Mach number levels from the inviscid/viscous and Navier-Stokes solutions. There are some distinct differences in the computation when pure inviscid flow is assumed. Without viscous effects, the analysis predicts a more sudden jump in flow properties across the shock whose location is shifted more rearward than that predicted by the viscous solutions.

This is clearly shown in the plots of the surface static pressure distributions in figure 4.23 where the results are superimposed.

Figure 4.23: Comparisons of Computed Surface Static Pressure: Inviscid, Inviscid/Viscous and Viscous Solutions
Subramanian and Bozzola (1987) made a similar study to compare the inviscid and full viscous solutions along 70% blade height of the blade. They observed similar differences in the prediction of the location and the jump in properties across the shock when the Mach number distributions along mid-passage of the cascade are compared.

The observations thus emphasize the importance of including viscous modelling to accurately capture the physics of the flow near the shock. The ability to predict the shock location accurately, and to include the effect of shock smearing and shock-boundary-layer interaction, becomes important in blade design attempts. In this case, even the simple viscous modelling is shown to improve the results significantly.

Chapters 6 and 7 describe the attempts carried out to redesign the blade. Viscous shock phenomena predominate in the flow field of this test case. Thus design attempts are made to consider the effect of geometrical changes on the strength and location of the shock formation.
CHAPTER 5
THEORY OF THE DESIGN METHODS

This chapter is concerned with the development of the two design methods. The two methods are designated Method I and Method II and they differ primarily in their design specifications. Method I uses the mass-averaged swirl velocity, \( r\vec{V}_\theta \), and Method II is based on the specification of the distribution of blade pressure difference, \( \Delta P \).

Sections 5.1 and 5.2 give the details of the two methods.

5.1 THEORY OF METHOD I: SPECIFICATION OF \( r\vec{V}_\theta \)

As stated in Chapters 1 and 2, swirl velocity has been successfully used for many 3-D applications. In particular, it is used in the method of Zangeneh (1991) for a wide range of turbomachinery applications (Goto\(^1\) and Zangeneh, 1998, Zangeneh, 1998, Watanabe\(^1\) and Harada\(^1\), 1999).

The in-house experience acquired in its use has reached a level of proficiency where optimum designs are now readily achieved (for example, Ashihara\(^1\) and Goto\(^1\), 1999, Yiu and Zangeneh, 1998b, Zangeneh et al., 1998), thus making the development of the present method especially attractive.

---

\(^1\) Research and Design Engineers at Center for Fluids and Mechanical Engineering of Ebara Research Company, Limited (industrial sponsor of the project)
The background theory of the current development is similar to that used by Zangeneh (1991) (and therefore, Hawthorne et al., 1984, Tan et al., 1984 and Soulis, 1985) but the present method differs from previous versions in three main ways,

1) *It can be applied to blades in subsonic, transonic and supersonic flow; previous methods are based on potential flow and the flow is assumed to be subsonic (except in the method of Soulis).*

2) *Viscous effects are included; previous methods assume the flow to be purely inviscid.*

3) *It is applied to design blades with finite thickness; previous methods can only be applied to blades having zero thickness.*

The current method is restricted only by the assumptions made in the flow computation (i.e. the Euler solver). These have been given in Chapter 3, section 3.2. The method has been developed in both two and three dimensions. In the next section, the basic procedure of the method is described. The background theory, common to the methods of Hawthorne et al. (1984), Tan et al. (1984), Soulis (1985) and Zangeneh (1991) then follows. With the foundation described, the new blade design algorithm is presented. The algebraic formulation of the design algorithm is first carried out in two dimensions in rectilinear coordinates, and then presented in three-dimensional cylindrical polar coordinates. Finally, the methods used to generate the design specification and to approximate an initial blade are described.

### 5.1.1 Description of the Procedure

This inverse design procedure consists of two integrated parts. The first is the flow solver and the second is the blade update algorithm that calculates the necessary geometrical modification.

The procedure involves coupling the solver code and the blade design algorithm together to compute for the final blade shape. The computation process is iterative, whereby the blade modifications at each design iteration are derived directly from the
difference between the current mass-averaged swirl velocity distribution and the specified target distribution.

The method presumes that the preliminary configurations of the turbomachine have already been decided so that the design of the blade or cascade geometry can commence; that is, it already has an approximate definition of the flow-path, the required specific work, the aspect ratios and cascade solidity. It is also presumed that the flow rate has been specified at the outlet based on the cycle requirements, and that a rotational (or wheel) speed has also been chosen based upon structural considerations. These are necessary inputs to the design procedure whose basic steps are given in the flowchart of figure 5.1.

![Figure 5.1: Simplified Flow Chart of Design Method I](image-url)
In the procedure, the Euler solver described in Chapter 3 is used directly without any necessary modifications and there is no difference in the code used for design or for a single analysis run. This is an advantage of the current method. Some of the existing techniques (and in fact, the second method) require the modification of the analysis code when it is used for design, for example, to model the blade wall surfaces as permeable using the Transpiration model, as described in Chapter 2.

The flow solution and blade calculation therefore converge simultaneously to give the final blade geometry and the corresponding steady state flow solution.

5.1.2 Background Theory

As stated earlier, the prescribed quantity in this method is the mass-averaged swirl velocity, $r\vec{V}_\theta$. This is defined as,

$$r\vec{V}_\theta = \frac{\int_{US}^{LS} rV_\theta (\rho V_m) \, d\theta}{\int_{LS}^{US} (\rho V_m) \, d\theta}$$  (5.1)

where the integral is made in the tangential (or pitch-wise) direction from the upper surface, $US$ of one blade to the lower surface, $LS$ of the next blade in the cascade. The specification of $r\vec{V}_\theta$ is not arbitrary but is related to the specific work, $\dot{W}_T$, required by the machine through the turning of its blades. This relation is given by the Euler equation of turbomachinery, which states,

$$\dot{W}_T = \omega (rV_{\theta_{TE}} - rV_{\theta_{LE}})$$  (5.2)

where $\omega$ is the rotational speed, $r$ is the radius and $V_{\theta_{TE}}$ and $V_{\theta_{LE}}$ are the tangential velocities at the trailing- and leading-edges respectively.
The rate of change of the averaged swirl velocity along the blade is also related to the surface static pressure difference, \( P^+ - P^- \) across the blade where the superscripts + and - denote the upper and lower surfaces of the blade respectively. This is shown by Hawthorne et al. (1984) using the assumption of an incompressible and irrotational flow for a thin blade, to be

\[
P^+ - P^- = \frac{2\pi}{N} \rho V_{m\phi} \frac{\partial r \bar{\nu}_\phi}{\partial m}
\]

(5.3)

where \( V_{m\phi} \) is the averaged meridional velocity across the blade and \( \frac{\partial r \bar{\nu}_\phi}{\partial m} \) is the meridional derivative of the tangentially averaged swirl (in the present method, the mass-averaged value is used and its meridional derivative, \( \frac{\partial r \nu_{\phi}}{\partial m} \) is referred to as the \textit{Blade Loading}).

These considerations indicate that the advantages in the use of the swirl velocity, \( r \bar{\nu}_\phi \) as the design parameter are:

1) It allows design to be carried out directly based on the required work done;
2) It allows control over the pressure distribution of the blade; and moreover,
3) It allows the specification of blade thickness.

It remains to be shown how the target swirl, \( r \bar{\nu}_{\phi}^* \) can be used to design a blade geometry. Following previous work but extending the terminology for a thick blade, the blade surfaces, \( \alpha^\pm \) can be written as,

\[
\alpha^\pm = \theta - \left( f \pm \frac{t_\theta}{2} \right) = n \frac{2\pi}{N} \quad n=1,2,3,\ldots,N
\]

(5.4)

where \( f \) is the mean camber line, \( t_\theta \) (see figure 5.2) is the tangential thickness, and where \( f \pm \frac{t_\theta}{2} \) represents the angular coordinates of points on the upper and lower
surfaces of the blade, $\theta$ is the tangential coordinate of a cylindrical polar coordinate system and $N$ is the number of blades in the cascade arrangement.

Figure 5.2: Blade Cascade with Prescribed Tangential Thickness

With this convention, the thickness of a blade can be based on a specified uncambered aerofoil and set off on the camber line at each chordal station to form the blade. This is in agreement with current design practice. The next two sections describe the development of the blade update algorithm in two and three dimensions respectively.
5.1.3 Derivation of the Camber Line Update Algorithm:
Two Dimensions

The design method is first developed in two dimensions where the implementation of
the method is tested before the idea is extended to three dimensions with no real change
in "strategy".

In two dimensions, the mass-averaged swirl velocity is equivalent to the mass-averaged
tangential velocity. Thus, equation (5.1) restated in two-dimensional rectilinear
coordinates (x,y) becomes,

\[ \bar{V}_y = \frac{\int_{LS}^{US} V_y (\rho V_x) dy}{\int_{LS}^{US} (\rho V_x) dy} \quad (5.5) \]

Using equation (5.4), the blade surfaces represented in (x,y) are,

\[ \alpha^a = y - \left( f \pm \frac{t_a}{2} \right) = ns \quad \text{where } n=1,2,3,..., N \quad (5.6) \]

where \( y \) represents the tangential coordinates, \( f \) is the camber line and \( s \) is the blade
pitch in metres.

In the flow computation, the surface flow is allowed to slip along the solid walls of the
blade surfaces. Mathematically, the tangency condition can be expressed as,

\[ W^\pm \nabla \alpha^a = 0 \quad (5.7) \]

where \( \nabla \alpha^a \) represents the vectors normal to the blade surfaces and \( W^\pm \) are the relative
velocities at the upper and lower blade surfaces.
Expanding equation (5.6), the vector product gives,

\[ \frac{d\left(f \pm \frac{t_0}{2}\right)}{dx} V_x^\pm = V_y^\pm - U \]  

(5.8)

where \( U \) represents the blade speed in the 2-D flow field. Applying the equations to an initial camber, \( f^0 \) and adding the two equations shown in equation (5.8) then gives,

\[ \left(V_x^+ + V_x^-\right) \frac{df^0}{dx} + \left(V_y^+ - V_y^-\right) \frac{dt_0}{dx} = \left(V_y^+ + V_y^-\right) - 2U \]  

(5.9)

where superscript 0 corresponds to the flow field computed for the blade shape defined with the initial camber, \( f^0 \).

Defining the blade average value with subscript \( bl \) and the blade difference value with notation \( \Delta \), the terms in equation (5.9) are represented as follows,

\[ V_{x_{bl}} = \frac{1}{2} \left(V_x^+ + V_x^-\right) \quad \& \quad V_{y_{bl}} = \frac{1}{2} \left(V_y^+ + V_y^-\right) \]

\[ \Delta V_x = \left(V_x^+ - V_x^-\right) \]  

(5.10)

which, substituted into equation (5.9) gives,

\[ V_{x_{bl}}^0 \frac{df^0}{dx} + \frac{1}{4} \frac{dt_0}{dx} \Delta V_x^0 = V_{y_{bl}}^0 - U \]  

(5.11)

So far, the algebra concerns only the tangency condition of the surface flow which is satisfied by the solid wall boundary condition in the analysis code. Nonetheless, this forms the basis for the blade update algorithm.
For the design problem, a correction term is introduced in the r.h.s. of the equation to signal a change in the blade camber. As the objective of the algorithm is to achieve the target swirl velocity, the correction term is defined as the difference between the target and the current values.

The geometrical modification is therefore made directly proportional to the design discrepancy (i.e. $\bar{V}_y^* - \bar{V}_y^0$) and the first approximation for the new geometry, $f^*$ can be made based on the current flow results (i.e. $V_{x,v}^0$, $\Delta V_{x}^0$ and $V_{y,v}^0$) viz.,

$$V_{x,v}^0 \frac{df^*}{dx} + \frac{1}{4} \frac{dt_0}{dx} \Delta V_{x}^0 = V_{y,v}^0 - U + [\bar{V}_y^* - \bar{V}_y^0]$$

(5.12)

Using this, the new camber geometry, $f^*$ can then be calculated. Alternatively but equivalently, the blade modification, $f^* - f^0$ (or $\Delta f^*$) can be computed instead. This is obtained by subtracting equation (5.11) from equation (5.12) to yield,

$$V_{x,v}^0 \frac{d(\Delta f^*)}{dx} = \bar{V}_y^* - \bar{V}_y^0$$

(5.13)

Thus, at the $(n+1)^{th}$ design update, the expression becomes,

$$V_{x,v}^n \frac{d(\Delta f^{n+1})}{dx} = \bar{V}_y^* - \bar{V}_y^n$$

(5.14)

Therefore, when the iterative process finally converges, the correction term on the r.h.s. of the equation (5.14) diminishes towards zero, thus giving no change to the current geometry. Mathematically, the progressive blade update sequence may be represented as follows,
Progressive Blade Update

**Known:** Estimate/input an initial blade camber, \( f^0 \) and specify the target mass-averaged swirl velocity distribution, \( \overline{V}_y^* \).

**Aim:** Determine the blade shape such that \( \overline{V}_y^* \) is achieved

**Algorithm:** Starting from \( f_0 \rightarrow \) Compute flow field, 0

Modify blade camber: \( f^l = f^0 + \left( \frac{\overline{V}_{y}^* - \overline{V}_{y}^0}{V_{x}^0} \right) \Delta x \)

Compute flow field, 1 and determine \( \overline{V}_{y}^l \)

Then determine new camber: \( f^2 = f^l + \left( \frac{\overline{V}_{y}^* - \overline{V}_{y}^l}{V_{x}^l} \right) \Delta x \)

**Iterative Result:** At \((n+1)\)th iteration, \( f^{n+1} = f^0 + \sum_{i=0}^{n} \left( \frac{\overline{V}_{y}^* - \overline{V}_{y}^i}{V_{x}^i} \right) \Delta x \)

**Convergence:** \( f^{n+1} \rightarrow f^* \) as \( \overline{V}_{y}^{n+1} \rightarrow \overline{V}_y^* \)

**5.1.3.1 Initial Condition: Fixed Stacking Point**

At each design iteration, equation (5.14) is integrated numerically to determine the new camber geometry, \( f^{n+1} \). The integration, as in the case of other initial value problems, cannot be completed without some initial condition on \( Af^{n+1} \) (\( = f^{n+1} - f^n \)). This initial value is given by the stacking condition of the blade. The stacking condition is implemented by giving as an input the value of the blade camber, \( f \) along any chosen axial position, for example, near the blade leading-edge. The stacking point then remains fixed throughout the iterative design procedure. The initial value of \( Af^{n+1} \) is therefore necessarily zero at the stacking position and the integration can proceed upstream and downstream of the fixed stacking point using the trapezoidal rule (see figure 5.3).
Figure 5.3: Stacking Condition: A Fixed Point along the Camber Line

Once the camber, $f^{n+1}$ is obtained, the specified tangential thickness is added symmetrically to the new camber to give the new blade geometry. In the current implementation, the design procedure is assumed to have converged when the current $\vec{v}_y^n$ matches the target value, $\vec{v}_y^*$ to within a specified tolerance.
5.1.4 Derivation of the Camber Line Update Algorithm:

Three Dimensions

In three dimensions, the derivation is carried out in cylindrical polar coordinates based on the same principle. Using the expression given by equation (5.4) to represent the blade in \((r, \theta, z)\), the tangency condition of the surface flow given by equation (5.8) in 3-D is,

Upper Surface: \( V_z^+ \frac{\partial}{\partial z} \left( \frac{f + \frac{t_\theta}{2}}{2} \right) = \frac{V_{\theta}^+}{r^+} - V_r^+ \frac{\partial}{\partial r} \left( \frac{f + \frac{t_\theta}{2}}{2} \right) - \omega \)  

(5.15)

Lower Surface: \( V_z^- \frac{\partial}{\partial z} \left( \frac{f - \frac{t_\theta}{2}}{2} \right) = \frac{V_{\theta}^-}{r^-} - V_r^- \frac{\partial}{\partial r} \left( \frac{f - \frac{t_\theta}{2}}{2} \right) - \omega \)  

(5.16)

where \(\omega\) is the rotational speed.

Following the same step as that used to derive the expression in two dimensions, equations (5.15) and (5.16) are applied to an initial camber, \(f^0\) and then combined to yield,

\[
V_{\omega} \frac{\partial f^0}{\partial z} + \frac{1}{4} \Delta V_z \frac{\partial}{\partial z} \left( \frac{f^0}{r^2} \right) + \frac{1}{4} \Delta V_{\theta} \frac{\partial}{\partial \theta} + \frac{1}{4} \Delta V_r \frac{\partial}{\partial r} - \omega
\]

where as before, \(V_{bl} = \frac{V^+ + V^-}{2}\) and \(\Delta V = V^+ - V^-\). Also, since \(r^+ = r^-\) in the vertical H-grid, the symbol \(r\) is used to represent the radius terms on both the upper and lower blade surfaces.

For the final blade geometry, the expression must satisfy the target specification. The final expression is therefore,

\[
V_{\omega} \frac{\partial f^*}{\partial z} + \frac{1}{4} \Delta V_z \frac{\partial}{\partial z} \left( \frac{f^*}{r^2} \right) + \frac{1}{4} \Delta V_{\theta} \frac{\partial}{\partial \theta} + \frac{1}{4} \Delta V_r \frac{\partial}{\partial r} - \omega
\]

(5.18)
The first approximation for the new camber, $f'$, is then,

$$V_{z u} \frac{\partial f'}{\partial z} + \frac{1}{4} \Delta V_{z} \frac{\partial \theta}{\partial z} = \frac{r \dot{V}_{\theta}^{o} + r \ddot{V}_{\theta}^{o} - r \dddot{V}_{\theta}^{o}}{r^2} - \frac{1}{4} \Delta V_{r} \frac{\partial \theta}{\partial r} - \frac{1}{4} \Delta V_{\theta} \frac{\partial \theta}{\partial \theta} - \omega$$  \hspace{1cm} (5.19)

Then, by subtracting equation (5.17) from (5.19), the necessary blade modification can be calculated from the solution of the resulting first-order partial differential equation, which is,

$$V_{z u} \frac{\partial (f' - f^{o})}{\partial z} + V_{n u} \frac{\partial (f' - f^{o})}{\partial r} = \frac{r \dot{V}_{\theta}^{o} - r \dddot{V}_{\theta}^{o}}{r^2}$$  \hspace{1cm} (5.20)

This equation is similar to the version used by Soulis (1985) in potential flow. The essential differences are that the earlier version considers only blades with zero thickness and has only the axial differential term (i.e. the second term on the left hand side of equation 5.20, was ignored).

It is unclear why Soulis omitted the radial differential term. One reason might be that, by doing so, the equation can be readily solved using quadrature.

Here, equation (5.20) is solved for the necessary blade changes. At the $(n+1)^{th}$ iteration and denoting the blade change to be $Af^{n+1} = f^{n+1} - f^{n}$, equation (5.20) becomes,

$$V_{z u} \frac{\partial Af^{n+1}}{\partial z} + V_{n u} \frac{\partial Af^{n+1}}{\partial r} = \frac{r \dot{V}_{\theta}^{o} - r \dddot{V}_{\theta}^{o}}{r^2}$$  \hspace{1cm} (5.21)

This is the final form of the blade update algorithm.
5.1.4.1 Initial Condition: Fixed Stacking Line

The required blade modification is calculated by integrating equation (5.21) along the meridional projections of streamlines on the blade surface. As with the solution of the equation derived in two dimensions (i.e. equation 5.14), the integration requires the input of an initial condition on $Af^{n+1}$ given by the stacking condition of the blade. In three dimensions, the stacking condition is the fixed blade wrap/camber angle, $f$ along a chosen quasi-orthogonal between the blade leading- and trailing-edges. In actual design task, the stacking position may be decided based on mechanical and/or vibrational considerations. The integration can then proceed throughout the meridional plane of quasi-streamlines and quasi-orthogonals, starting from the stacking line (see figure 5.4).

Figure 5.4: Physical Meridional Mesh (Showing also the Stacking Line) Mapped into the Rectangular "Computational" Mesh
5.1.4.2 Coordinate Transformation

To simplify the solution process with respect to the application of boundary conditions along the quasi-streamlines of the hub and shroud, the meridional mesh is mapped onto a rectangular "computational" mesh using a coordinate transformation technique (Thompson et al. 1985).

The transformation is performed on the meridional \((r, z)\) plane into the body-fitted \((\xi, \eta)\) plane. That is,

\[
(r, z) \rightarrow (\xi, \eta)
\]

such that the lines of \(\xi=\text{constant}\) and \(\eta=\text{constant}\) lie along the quasi-streamlines and quasi-orthogonals of the meridional mesh respectively (see figure 5.4).

The transformation parameters are defined by the Jacobian, \(J\) where,

\[
J = \left| \frac{\partial (r, z)}{\partial (\xi, \eta)} \right| = r_\xi z_\eta - r_\eta z_\xi
\]

(5.22)

and

\[
\begin{align*}
    r_\xi &= \frac{\partial r}{\partial \xi} \quad , \quad z_\xi = \frac{\partial z}{\partial \xi} \\
    r_\eta &= \frac{\partial r}{\partial \eta} \quad , \quad z_\eta = \frac{\partial z}{\partial \eta}
\end{align*}
\]

It can be shown that for any variable \(V(r, z)\) (see Appendix IV),

\[
\frac{\partial F}{\partial r} = \left(z_\eta F_\xi - z_\xi F_\eta \right)/ J \quad (5.23)
\]

\[
\frac{\partial F}{\partial z} = \left(r_\xi F_\eta - r_\eta F_\xi \right)/ J \quad (5.24)
\]
Applying the transformation to equation (5.21), the equivalent equation in the transformed \((\xi, \eta)\) plane is,

\[
V^n_{\xi} \left( \frac{r^n_\xi \Delta f^{n+1}_\xi - r^n_\eta \Delta f^{n+1}_\eta}{J} \right) + V^n_{\eta} \left( \frac{z^n_\eta \Delta f^{n+1}_\eta - z^n_\xi \Delta f^{n+1}_\xi}{J} \right) = \frac{r \tilde{V}^*_{\theta} - r \tilde{V}^n_{\theta}}{r^2}
\]

\[
\Rightarrow \frac{1}{J} \left( \left( V^n_{\xi} z^n_\eta - V^n_{\eta} r^n_\eta \right) \Delta f^{n+1}_\xi + \left( V^n_{\eta} r^n_\xi - V^n_{\xi} z^n_\xi \right) \Delta f^{n+1}_\eta \right) = \frac{r \tilde{V}^*_{\theta} - r \tilde{V}^n_{\theta}}{r^2} \tag{5.25}
\]

In the transformed plane, the velocity components in the \(\xi\) and \(\eta\) directions are,

\[
V^n_{\xi} = \left( V^n_{\xi} z^n_\eta - V^n_{\eta} r^n_\eta \right) \quad \& \quad V^n_{\eta} = \left( V^n_{\eta} r^n_\xi - V^n_{\xi} z^n_\xi \right) \tag{5.26}
\]

where velocities, \(V^n_{\xi}\), \(V^n_{\eta}\) are vertical and horizontal velocity components respectively in the \((\xi, \eta)\) plane.

Substituting equation (5.26) into equation (5.25) and rearranging, the blade update algorithm becomes,

\[
\left( \Delta f^{n+1}_\eta \hat{V}^n_{\eta} \over \hat{V}^n_{\xi} \right) = \frac{J}{V^n_{\xi}} \left( \frac{r \tilde{V}^*_{\theta} - r \tilde{V}^n_{\theta}}{r^2} \right) \tag{5.27}
\]

Equation (5.27) is then discretized using the Crank-Nicholson finite difference method and the discretized system of equations represents a tridiagonal matrix which is then solved using the Thomas algorithm (Press et al., 1992). Details of the discretization are given in Appendix V.

As in the 2-D procedure, the design process is completed when the current \(r \tilde{V}^n_{\theta}\) is achieved to within a specified tolerance of the target value. The way in which the target distribution is defined is given in the next section.
5.1.5 Three-Segment Method of Defining the Target Distributions

A versatile way of defining the target design distribution is adopted from Zangeneh et al. (1996) in the current implementation.

As advocated by Zangeneh, the basic approach to specifying the design specification in 3-D is to select optimum distributions of $r\vec{V}_\theta$ at the hub and shroud and linearly interpolate the values to generate the overall distribution on the whole of the meridional plane.

The values of $r\vec{V}_\theta$ at the leading-edge (L.E.) and trailing-edge (T.E.) are obtained from the designated velocity triangles. In addition, since it is known from equation (5.3) that the jump in pressure across the blade is given by

$$P^+ - P^- = \frac{2\pi}{N} \rho V_{n_c} \frac{\partial r\vec{V}_\theta}{\partial m}$$

(5.28)

the derivative of $r\vec{V}_\theta$ in the meridional direction at the L.E. may be used to control the flow incidence and set to zero at the T.E. to satisfy the Kutta condition.

The equation also shows the relation of the surface pressure to the $\frac{\partial r\vec{V}_\theta}{\partial m}$ (or loading distribution), therefore by specifying a smooth $\frac{\partial r\vec{V}_\theta}{\partial m}$ distribution, it is also possible to ensure smooth surface pressure variations. All these criteria are found to be more easily satisfied by specifying the loading distribution rather than the $r\vec{V}_\theta$ distribution.

Using the method of Zangeneh, the distribution for the meridional derivative of $r\vec{V}_\theta$ is divided into three sections as shown in figure 5.5.
The first segment is parabolic up to the point (NC), which is followed by a linear variation between the points NC and ND with a specified slope and finally by a second parabolic distribution which reduces the loading down to zero at the trailing-edge (NB) to satisfy the Kutta condition.

The target $rV_{\theta}$ distribution is then found by integrating these values of $\frac{\partial rV_{\theta}}{\partial m}$ (more details may be found in Appendix VII). Numerous types of loading characteristics can be described using the current method. Fore-loaded and aft-loaded distributions generated using this method are shown in figure 5.6.

**Figure 5.5: Three-Segment Specification**

**Figure 5.6: Fore- and Aft-Loaded Distributions**
5.1.6 Predicting an Initial Blade Camber Geometry

From the description of the procedure in section 5.1.1, the design process requires the input of an initial blade camber line, $f^0$, to initiate the procedure.

In most circumstances, design tasks are undertaken to improve the performance of a known blade or to create a new design based on an earlier design, which operates under similar conditions. In these circumstances, the initial blade camber will be known.

However, in cases where the initial camber geometry is not known (for example, in a completely new design), the geometry needs to be approximated. Here, the approximation is carried out using the specified mass-averaged swirl velocity distribution.

The approximation is made using equations (5.29) and (5.30) which are simple equations that assume that the blade camber is tangential to the mass-averaged relative velocity vectors.

\[ \text{In 2D:} \quad \frac{\partial f^0}{\partial x} = \frac{\partial f^0}{\partial y} - U \quad (5.29) \]

\[ \text{& In 3D:} \quad \frac{\partial f^0}{\partial z} = \frac{r \partial V^*}{r^2} - \omega \quad (5.30) \]

In the absence of a known flow-field, $\tilde{V}^*$ and $\tilde{V}^*$ can only be approximations of the mass-averaged axial velocity (given as a constant or based on one-dimensional estimation) and the equations are solved to give a quick estimate of the initial camber geometry.

It is accepted that the prediction of the initial camber is highly simplified. Nevertheless, it need only be a rough estimate of the initial shape and the current method of approximation has been found to be adequate to start the design process in all of the cases encountered.
5.1.7 Design Convergence Criteria

As stated in section 5.1.1, the geometrical modifications of the blade are carried out after the convergence of each successive flow analysis using the solver code. The convergence criteria set for the flow calculation in the design procedure are identical to those used in a single analysis run which is given in Chapter 3, section 3.9. For clarity, the convergence conditions are listed here again.

1. Mass flow rate at each stream-wise grid line agrees with the inlet mass flow to within 0.5 percent.

2. The maximum change in the absolute velocity anywhere in the flow field divided by the root-mean-square of the absolute velocities in the flow field is smaller than $1.5 \times 10^{-5}$.

3. Inlet mass flow does not change by more than 0.1 percent over 50 time steps.

In the design procedure, less stringent criteria can be used in the flow analysis step to reduce the amount of computation required in the flow analysis and thus speed up the design process, without significantly compromising the accuracy of the design outcome. This is shown in a sample calculation in Chapter 6, section 6.1.2.1. Typically, the following settings have been found to be adequate when employing the solver code in the design procedure;

<table>
<thead>
<tr>
<th>Criterion 1: 1% and below</th>
</tr>
</thead>
<tbody>
<tr>
<td>Criterion 2: $5 \times 10^{-5}$ and below</td>
</tr>
<tr>
<td>Criterion 3: Check only in the final stage</td>
</tr>
</tbody>
</table>

The design process for the computation of the blade or cascade geometry is considered to have converged when the calculated mass-averaged swirl velocity matches the target distribution to within a specified tolerance.
A compromise between the computational cost and the tolerance within which the final shape must match the target specification has been accepted as inevitable. The use of 1% as the convergence criterion between the computed and target design mass-averaged swirl velocity specification is generally found to be adequate and is used in all the redesign attempts presented (sections 6.1.2, 6.2.2 and 6.2.3).

In validations of the method in two and three dimensions, however, the aim is to check that the design methods can reproduce the known geometry based on the specification of its mass-averaged swirl velocity. Thus, a stricter design criterion is used. In the two design validation cases which will follow in sections 6.1.1 and 6.2.1, the design convergence is set at 0.5 % maximum difference between the calculated and target swirl velocity distributions.

It is noted that in all the redesign attempts demonstrated using this method and the second method (section 5.2). The final mass flow rates are checked to be closely matching that in the original cascades.
5.2 THEORY OF METHOD II: SPECIFICATION OF $\Delta P$

This section is concerned with the theory of the second design method (designated *Method II*) in which the difference in blade surface static pressure (or $\Delta P$) is chosen as the design parameter.

The prescribed parameter, $\Delta P$ is explicitly imposed along the boundary of the blade surfaces using the transpiration model. This is the main difference from other inverse methods based on the transpiration technique, which use either the surface static pressure or the surface velocity distributions. As mentioned in the literature survey in Chapter 2, sections 2.3.2.2 and 2.3.2.4, the transpiration model replaces the blade solid wall boundary condition in the solver code with an artificial permeable surface model where flows are allowed to penetrate. Basically, if the target distribution is different from the one resulting from a direct calculation around the same geometry, the surface velocity will not be tangential to the walls, and in such cases the transpiring boundaries allow the convective fluxes to go through. The transpiring velocity then becomes the mechanism by which the blade updates are calculated.

As given in Chapter 2, the conventional choice of surface static pressure (or surface velocity) as the design specification has its failings in three dimensions. Although the use of surface pressure may be the most direct means by which to influence the surface flow (for example, to control flow diffusion), the design concept only applies to 2-D cases where it is assumed that the surface streamlines of the blade section lie in the two-dimensional plane under consideration. In actual turbomachinery applications, such an assumption is unrealistic as the SI stream-surfaces may twist and warp as they pass through the blade row. Prior to having the blade and performing a full 3-D calculation of its flow, an accurate estimation of the locations of the blade-surface streamlines is very difficult.

In addition, it is also difficult to prescribe the conventional choice of flow quantities in 3-D as the distributions at the hub and the tip are related and cannot be specified arbitrarily or independently. Any inconsistency in the prescription of these quantities has been known to result in convergence difficulties or non-feasible blades having open
or cross-over ends (i.e. negative thickness). Consequently, it is not clear whether there is an advantage in specifying the blade surface pressure distributions in 3-D flows (Dang, 1995).

By specifying the $\Delta P$ distribution together with the tangential thickness, it is possible to control the blade pressure loading directly, which is very important in controlling shock losses in transonic/supersonic flow and cavitation in incompressible flow, while maintaining structurally sound blades. Furthermore, the $\Delta P$ specification can be readily used in 3-D. However, one shortcoming of the $\Delta P$ specification is that, in some cases, it can be difficult to design exactly for the correct specific work and some iteration may be necessary.

Like Method 1, this method was first tested in two dimensions and extended to 3-D. In this instance, details of its extension to 3-D, which form the basis of originality for the current development, are presented. Other than the author's, similar design methods have also been given by Dang (1995) and Ahmadi and Ghaly (1997) in 2-D (see Chapter 2, section 2.3.2.4). Recently, Damle and Dang (1998) reported 3-D application of their method but no details of their formulation were given.

The basic steps involved in the present method are described in the next section, following which, the transpiration model based on $\Delta P$ is introduced. In sections 5.2.3 and 5.2.4, the blade design algorithms in two and three dimensions are presented.

It is noted that in the current proposed methods, the surface flow tangency condition is used to determine the new blade geometry. The underlying principle is therefore not easily extended to full viscous solutions using N-S equations when the no-slip condition is applied along the boundaries of the blade.
5.2.1 Description of the Procedure

This design procedure consists of three main parts. The first is the original Euler flow solver ("direct" mode), the second is the modified Euler flow solver based on the new transpiration model ("design" mode) and the third is the blade update algorithm.

The bulk of the computation is carried out in the modified solver and in performing the blade updates. In the "design" mode flow calculation, the presence of the blade in the flow field is represented by the imposed $\Delta P$ distribution. The blade update is carried out using the surface transpiring flow velocities such that its surfaces are realigned with the flow, thereby satisfying the tangency condition. As the iterative design procedure converges, the amount of transpiring normal flow decreases progressively, eventually giving the final blade shape. In the last step, the original flow solver (i.e. "direct" mode calculation) is used to give the final flow-field. The flow fields given by the solver in "design" and "direct" modes for the final blade design are very similar. The flow computation in "direct" mode is included here mainly to check the design by comparing the target $\Delta P$ distribution with that finally obtained.

Like the first method, the procedure requires that the preliminary configurations of the turbomachine have been defined up to the point where the required specific work, the aspect ratio, cascade solidity, as well as the specified mass flow and the rotational speed, are all known. The design of the blade can then commence using the proposed design method, which is described by the flow chart in figure 5.7.
**Inverse Design Method II**

1. **Initial Blade Geometry**
2. **Specify Surface Static Pressure Difference, \( \Delta P \)**
3. **Modified Euler Flow Analysis: Transpiration Model**
4. **Use Transpiring Flow to Modify the Blade Geometry: Surface Tangency Condition**
5. **Is the Calculated Blade Modification Sufficiently Small?**
   - **No**
   - **Yes**
7. **Euler Flow Analysis: Original Solver**
8. **Final Inverse Designed Blade and its Flow**

**Input Geometrical Parameters:**
- Cascade solidity
- Domain axial coordinate
- Tangential thickness
- Stacking location

**Additional for 3D:**
- Hub and Casing surfaces
- Domain radial coordinate

**Flow Conditions:**
- Boundary Conditions for Solver
- Rotational Speed

*Figure 5.7: Simplified Flow Chart of Design Method II*
5.2.2 Transpiration Model

In the beginning of section 5.2, it is mentioned that in the transpiration model, the surface static pressure difference distribution is imposed directly along the blade surfaces. This parameter is given as the difference between the pressures on the upper and lower surfaces of the blade; that is,

$$\Delta P = P^+ - P^-$$  \hspace{1cm} (5.31)

In the following two sub-sections, the implementation of the permeable wall model using $\Delta P$ is shown.

5.2.2.1 Modification of the Euler Solver: Two Dimensions

In the modified Euler solver, the blade surface static pressures are modified after each time step to ensure that their difference across the blade surface is the target value. Denoting the target quantity as $\Delta P^*$, the surface pressures are updated as follows,

$$(P^z)^{m+1} = (P_{bl})^m \pm \frac{1}{2} \Delta P^*$$  \hspace{1cm} (5.32)

where $(P^z)^{m+1}$ are the updated upper and lower surface pressures and $(P_{bl})^m$ denotes the original value of the blade-averaged pressure defined as,

$$(P_{bl})^m = \frac{1}{2} (P^+ + P^-)^m$$  \hspace{1cm} (5.33)

with the superscripts, $m$ and $m+1$ corresponding respectively to the values before and after the target $\Delta P$ is imposed.
In addition to the surface static pressure, the surface tangential velocities are also updated. This is performed, using,

\[
(V_y)^{m+1} = (V_y)^m \pm \frac{I}{2} \Delta V_{y_{frac}}
\]  

(5.34)

where \((V_y)^m\) is the current blade-averaged value (i.e. \(= \frac{I}{2} \left[(V_y^+)^m + (V_y^-)^m\right]\)). \(\Delta V_{y_{frac}}\) in this equation represents the difference between the tangential velocities on the upper and lower surfaces of the blade section. It is calculated based on satisfying the tangency condition using the computed value of the axial velocities, \(V_x^\pm\) and the current blade surface geometries. It is obtained by subtracting the expressions given in equation (5.8), which gives,

\[
\Delta V_{y_{frac}} = V_{y_{frac}}^+ - V_{y_{frac}}^-
\]

\[
= (V_x^+ - V_x^-) \frac{df}{dx} + \frac{l}{2} (V_x^+ + V_x^-) \frac{dt_\theta}{dx}
\]

(5.35)

Equations (5.32) and (5.34) are also given by Dang (1995) and Ahmadi and Ghaly (1997) in their two-dimensional procedures and have been shown to give added robustness to the scheme.

It is noted that when the design procedure finally converges, the target blade pressure difference is fully satisfied and the flow is aligned with the surfaces of the final blade. Equations (5.32) and (5.34) then provide no change to the computed surface pressures and tangential velocities respectively.
5.2.2.2 Modification of the Euler Solver: Three Dimensions

Implementation of the permeable wall boundary condition in three dimensions is presented here. The formulation is carried out in a similar manner to that in two dimensions except that it is made in the cylindrical polar coordinates \((r, \theta, z)\).

The specification of \(\Delta P^*\) follows equation (5.32) and the tangential velocity is also updated using the blade-difference values computed by satisfying the tangency condition. The surface flow tangency condition in 3-D written separately for the upper and lower surfaces of the blade is given in equations (5.15) and (5.16), which are

\[
\text{Upper Surface:} \quad V_z^+ \frac{\partial f^+}{\partial z} = V_{\theta^+}^+ r^+ \frac{\partial f^+}{\partial r} - \omega \quad (5.36)
\]

\[
\text{Lower Surface:} \quad V_z^- \frac{\partial f^-}{\partial z} = V_{\theta^-}^- r^- \frac{\partial f^-}{\partial r} - \omega \quad (5.37)
\]

Subtracting equation (5.37) from equation (5.36) and rearranging then gives the blade-difference value, \(\Delta V_{\theta_{\text{upl}}}\) as,

\[
\Delta V_{\theta_{\text{upl}}} = \left( r^+ V_z^+ \frac{\partial f^+}{\partial z} + r^+ V_r^+ \frac{\partial f^+}{\partial r} \right) - \left( r^- V_z^- \frac{\partial f^-}{\partial z} + r^- V_r^- \frac{\partial f^-}{\partial r} \right) \quad (5.38)
\]

which is evaluated on the transformed \((\xi, \eta)\) plane described earlier in section 5.1.4.2.

This value is then used in the same way as in two dimensions to update the surface tangential velocities which are,

\[
\left( V_{\theta^i} \right)^{m+1} = \left( V_{\theta^i} \right)^m \pm \frac{1}{2} \Delta V_{\theta_{\text{upl}}} \quad (5.39)
\]

where \(\left( V_{\theta^i} \right)^m\) is the blade-averaged value of the transpiring tangential velocity.
5.2.3 Derivation of the Camber Line Update Algorithm: Two Dimensions

Here, it is shown how the computed transpiring flow velocity is used to update the blade shape. Computation of the new blade (or blade section) shape is performed at the end of each flow computation using the modified solver. The blade is updated by adjusting the camber shape such that the new blade surfaces are realigned with the transpiring flow, using the tangency flow condition. Following equation (5.8), the condition satisfied in two dimensions can be expressed as follows,

\[ \frac{d}{dx} \left( f + \frac{t_g}{2} \right) = \frac{(V_y^+ - U)}{V_x^+} \quad (5.40) \]

Upper Surface:

\[ \frac{d}{dx} \left( f - \frac{t_g}{2} \right) = \frac{(V_y^- - U)}{V_x^-} \quad (5.41) \]

Lower Surface:

Denoting the flow quantities of the current blade with the superscript \( n \), it follows that the new blade camber is determined by applying equations (5.40) and (5.41) (evaluated with the current values of the transpiring velocity components), and adding to give,

\[ \frac{df^{n+1}}{dx} = \frac{1}{2} \left[ \left( \frac{V_y^+ - U}{V_x^+} \right)^n + \left( \frac{V_y^- - U}{V_x^-} \right)^n \right] \quad (5.42) \]

where \( f^{n+1} \) is the new camber geometry of the blade whose surfaces are parallel to the surface transpiring velocity vectors.

Equation (5.42) is solved numerically to yield the new camber geometry, \( f^{n+1} \). As in Method I, the integration requires the initial condition of \( f^{n+1} \). This is provided by the stacking point of the blade, which remains fixed (i.e. \( f_{stack}^{n+1} = f_{stack}^n \)) during the iterative design procedure and from which the integration, using the trapezoidal rule, can continue (see figure 5.3).
5.2.4 Derivation of the Camber Line Update Algorithm: Three Dimensions

In three dimensions, the blade update equation is derived in cylindrical polar coordinates. Representation of the blade in \((r, \theta, z)\) follows that used in section 5.1.2, equation (5.4). As in two dimensions, the new camber geometry, \(f_{n+1}^{n+1}\) is also determined by realigning the blade geometry with the transpiring flow. This is obtained by applying the transpiring velocity components with the slip conditions given in equations (5.36) and (5.37) and then adding to give,

\[
(V_z^+ + V_z^-)^n \frac{\partial f_{n+1}^{n+1}}{\partial z} + \left(\frac{V_z^+ - V_z^-}{2}\right)^n \frac{\partial \theta}{\partial z} + (V_r^+ + V_r^-)^n \frac{\partial f_{n+1}^{n+1}}{\partial r} + \left(\frac{V_r^+ - V_r^-}{2}\right)^n \frac{\partial \theta}{\partial r} = \left(V_{\theta_r}^+ + V_{\theta_r}^-\right)^n - 2\omega
\]

where the superscripts, \(n\) and \(n+1\) correspond to the current and new geometries respectively.

On putting the terms involving \(f_{n+1}^{n+1}\) on the l.h.s., using subscript \(bl\) to represent the blade-averaged value, notation \(\Delta\) to denote the blade-difference value and replacing \(r^+\) and \(r^-\) by \(r\) (since \(r^+ = r^-\) in the vertical H-grid topology), the final expression for the blade update algorithm becomes,

\[
V^n \frac{\partial f_{n+1}^{n+1}}{\partial z} + V^n \frac{\partial f_{n+1}^{n+1}}{\partial r} = \left(V^n_{\theta_r} - \omega\right) - \frac{1}{4} \Delta V^n \frac{\partial \theta}{\partial z} - \frac{1}{4} \Delta V^n \frac{\partial \theta}{\partial r}
\]

The initial condition required to perform the numerical integration is given by the fixed blade wrap/camber angle, \(f\) along a chosen quasi-orthogonal between the blade leading- and trailing-edges. As in Method I, the integration can then proceed throughout the meridional plane of quasi-streamlines and quasi-orthogonals, starting from the stacking line.
5.2.4.1 Coordinate Transformation

The numerical solution of the equation is obtained on the transformed body-fitted \((\xi, \eta)\) plane where the lines of \(\xi=\text{constant}\) and \(\eta=\text{constant}\) lie along the quasi-streamlines and quasi-orthogonals of the meridional mesh respectively (see figure 5.4, section 5.1.4.1). The transformation parameters used here are identical to those defined in section 5.1.4.2 and are applied to equation (5.44) to give,

\[
V^n_{\xi} \left( r_{\xi} f_{\eta}^{n+1} - r_{\eta} f_{\xi}^{n+1} \right) + V^n_{\eta} \left( z_{\eta} f_{\xi}^{n+1} - z_{\xi} f_{\eta}^{n+1} \right) = \left( \frac{V^n_{\theta_\xi}}{r} - \omega \right) - \frac{1}{4} \Delta V^n_{\xi} \left( \frac{r_{\xi} t_{\theta_\xi} - r_{\eta} t_{\theta_\eta}}{J} \right) - \frac{1}{4} \Delta V^n_{\eta} \left( \frac{z_{\eta} t_{\theta_\eta} - z_{\xi} t_{\theta_\xi}}{J} \right)
\]

Re-arranging then gives,

\[
\frac{1}{J} \left[ \left( V^n_{\eta} z_{\eta} - V^n_{\eta} r_{\eta} \right) f_{\eta}^{n+1} + \left( V^n_{\xi} r_{\xi} - V^n_{\xi} z_{\xi} \right) f_{\eta}^{n+1} \right] = \left( \frac{V^n_{\theta_\xi}}{r} - \omega \right) - \frac{1}{4J} \left[ \left( \Delta V^n_{\xi} z_{\eta} - \Delta V^n_{\eta} r_{\eta} \right) t_{\theta_\xi} + \left( \Delta V^n_{\xi} r_{\xi} - \Delta V^n_{\eta} z_{\xi} \right) t_{\theta_\eta} \right]
\]

The final expression for the blade update equation in the \((\xi, \eta)\) plane is then,

\[
f_{\eta}^{n+1} + \frac{V^n_{\xi}}{V^n_{\eta}} f_{\eta}^{n+1} = \frac{J}{V^n_{\eta}} \left( \frac{V^n_{\theta_\xi}}{r} - \omega \right) - \frac{1}{4V^n_{\eta}} \left( \Delta V^n_{\xi} t_{\theta_\xi} + \Delta V^n_{\eta} t_{\theta_\eta} \right)
\] (5.45)

where \(V^n_{\xi}\) and \(V^n_{\eta}\) are the vertical and horizontal velocity components respectively in the transformed plane, as given by equation (5.26) of section 5.1.4.2. The velocity terms are actual values along the surfaces of the blade, but it is noted that in the undesirable case when the design specification results in a large region of surface flow separation, equation (5.45) breaks down. In such cases, the program code warns the user of unsatisfactory design specification and seeks the nearest geometry by taking mass-averaged values of the horizontal component of the velocities (i.e. \(\bar{V}_{\eta}^n, \bar{V}_{\eta}^n\)) instead of
the original surface values (i.e. $\Delta V_{\eta}$, $V_{\eta_{c}}$). One such example is encountered in Chapter 7, section 7.2.1 where the design code is used to reproduce the original NASA rotor 67 which experiences a region of shock-induced separation along the suction surface near the tip of the blade.

In either the original or modified form, the blade update equations are discretized using the Crank-Nicholson finite difference method and the resulting system of equations is solved for the whole of the meridional plane to give the new camber shape, $f^{n+1}$. The details may be referred to in Appendix VI.

### 5.2.5 Defining the Target $\Delta P$ Distribution

The relationship between the pressure jump across the blade and the swirl velocity is given in equation (5.3), section 5.1.2. It is then clear that the local rate of change of the mass-averaged swirl, $r\tilde{V}_\theta$ is directly related to the blade pressure loading,

\[
\Delta P \propto \frac{\partial r\tilde{V}_\theta}{\partial m} \tag{5.46}
\]

or

\[
\int_{LE}^{TE} \Delta P \ dm = K\left[\left(r\tilde{V}_\theta\right)_{TE} - \left(r\tilde{V}_\theta\right)_{LE}\right] \tag{5.47}
\]

which states that the area under the $\Delta P$ versus $m$ curve is related to the specific work, quantified by the overall change in swirl between the inlet and outlet of the cascade, i.e. $\left[r\tilde{V}_{\theta_{LE}} - r\tilde{V}_{\theta_{TE}}\right]$ and $K$ is the constant of proportionality.

Since the constant of proportionality in equation (5.47) is not known in a viscous, compressible flow, the correct value of $\Delta P$ is obtained iteratively. This is carried out by first defining a $\Delta P$ distribution (for example, using the three-segment method, see Appendix VII) which, if necessary, is varied against the resulting overall change in swirl during the design process to give eventually the required specific work. This process
may be performed manually or automatically using an outer loop attached to the proposed method. Implementation of the latter is given in the next section.

5.2.5.1 Secant Loop: Design for Specific Work

The objective of having an outer loop system is to automatically adjust the target pressure loading distribution such that the required specific work, quantified by the change in the mass-averaged swirl velocity from the inlet to the outlet of the final converged cascade (i.e. the bracketed terms on the r.h.s. of equation 5.48), is satisfied at every stage of the design. To illustrate the principle of this outer loop, details are presented here in two dimensions.

In two dimensions, the proportional relation between $\Delta P$ and the change in swirl is given by

$$\int_{LE}^{TE} \Delta P \, dx = K \left[ \bar{V}_{y_{TE}} - \bar{V}_{y_{LE}} \right]$$

(5.48)

The idea is to vary the unknown constant of proportionality in equation (5.48) such that the required specific work (i.e. $\left[ \bar{V}_{y_{TE}} - \bar{V}_{y_{LE}} \right]$) is attained.

In the proposed method, the outer iterative loop is executed using a fast converging, root-seeking Secant Method (Press et al., 1992) at each blade update interval where the pressure loading distribution is also updated.

Although this method may be used with any $\Delta P$ distribution, for the purpose of illustration, the problem is simplified by assuming a parabolic $\Delta P$ distribution so that its integral, $\int_{LE}^{TE} \Delta P \, dx$ is directly related to $\Delta P_{\text{max}}$ at mid-chord.
The parabolic target pressure loading follows the relation,

\[ \Delta P^* \propto x(l - x) \quad L.E. \leq x \leq T.E. \quad (5.49) \]

where the Kutta condition is satisfied at the trailing-edge, with \( x \) denoting the axial coordinate from the leading-edge to the trailing-edge of the blade section whose axial chord is set at 1.

Denoting the peak pressure load at the mid-axial chord as \( \Delta P_{max}^* \), equation (5.49) can be written as,

\[ \Delta P^* = 4 \Delta P_{max}^* x(l - x) \quad (5.50) \]

To find the required peak pressure load, \( \Delta P^* \), the algorithm starts with an arbitrary solution (point 1) on the first iteration (see figure 5.8) and the next improvement is taken as the point where the approximating line crosses the solution axis (point 2). As the iteration proceeds, the solution continually improves. Finally, the required value of the peak pressure load, \( \Delta P_{max}^* \) is obtained which, when used to define the target pressure loading using equation (5.50), gives a specific work sufficiently close to that specified. Typical tolerance is set at 1% of the required change in swirl between the leading- and trailing-edges.

Figure 5.8: Secant Algorithm: Finding \( \Delta P \) for a Required Specific Work
Mathematically, the \textit{Secant algorithm} is expressed as

\[ \Delta P_{\text{sec}}^{N+1} = \Delta P_{\text{sec}}^N - \frac{ASI}{ASI - AS2} \left( \Delta P_{\text{sec}}^N - \Delta P_{\text{sec}}^{N-1} \right) \quad (5.51) \]

where \(ASI = \Delta \tilde{V}_{\text{YTE,LE}}^* - \Delta \tilde{V}_{\text{YTE,LE}}^N\) and \(AS2 = \Delta \tilde{V}_{\text{YTE,LE}}^* - \Delta \tilde{V}_{\text{YTE,LE}}^{N-1}\), with \(\Delta \tilde{V}_{\text{YTE,LE}}\) representing the change in the mass-averaged tangential velocity between the inlet and outlet of the cascade, and superscripts, \(N, N+1\) and \(^*\) denoting the previous, current predicted and target values respectively.

The working of this automatic looping system is demonstrated in Chapter 7, section 7.1.2.1.

\subsection*{5.2.6 \hspace{1cm} Predicting an Initial Blade Camber Geometry}

In the case when the initial blade or blade section is not known, the geometry has to be approximated. The approximation is carried out in the same way as that used in \textit{Method I} using equations (5.29) and (5.30) of section 5.1.6 where more details can be found.
5.2.7 Design Convergence Criteria

As described earlier, the main design modules consist of the modified Euler solver and the blade update algorithm. Every time the transpiring solver outputs its flow solution, the numerical procedure initiates a design iteration to change the blade shape. In the current implementation, the "design" mode flow calculation is allowed to converge before its solution is used for the blade modification. Numerical experimentation has shown that it is not necessary to achieve the same level of convergence as that required by the "direct" mode flow computation (see chapter 3, section 3.9) throughout the entire design process. Instead, it is found to be sufficient and computationally faster to relax the convergence requirements of the flow computation at the early stages of the design, when large geometrical modifications may be made quickly without affecting the final design.

In the current implementation, two convergence settings, Set 1 & Set 2 are used to determine when the blade geometry is updated. When the number of iterations, \( N_{\text{iteration}} \) exceeds the given maximum number of time steps, \( N_{\text{max}} \), the less stringent criteria, Set 2 are used so as to reduce computational time. This is given as follows,

\[
\begin{align*}
&\text{Set the Minimum Number of time steps in the "design mode" analysis: } N_{\text{min}} \\
&\text{Set the Maximum Number of time steps in the "design mode" analysis: } N_{\text{max}} \\
&\text{Set Convergence Criteria: Mass Error and } V_{\text{rms}} \text{ Error} \\
&\quad \text{Set 1: A Strict Setting} \\
&\quad \text{Set 2: A Less Stringent Setting} \\
\end{align*}
\]

Condition loop based on the number of iterations in the flow computation, \( N_{\text{iteration}} \):

If \( N_{\text{max}} > N_{\text{iteration}} > N_{\text{min}} \):

Continue analysis until the set 1 of convergence criteria is satisfied.

If \( N_{\text{iteration}} > N_{\text{max}} \):

Disregard set 1 of convergence criteria and switch to the less stringent set 2. 
Continue flow computation until set 2 of convergence criteria is satisfied.

Additional Check: Set 1 of the convergence criteria must be satisfied when the design procedure finally converges.
The convergence criteria given in Set 1 is chosen to be the same as for the "direct" mode computation (i.e. mass error ≤ 0.5% and \( V_{rms} \) error ≤ 1.5x10^{-5}) and Set 2 is chosen to have mass error ≤ 3.0% and \( V_{rms} \) error ≤ 10x10^{-5}. Typical values of \( N_{\text{min}} \) and \( N_{\text{max}} \) are 85 and 300 respectively. At advanced stages of the design process (i.e. when the blade shape is nearing convergence), Set 2 is not used and the flow computation is allowed to converge to the stricter criteria (i.e. Set 1). This is to ensure that the final design does not suffer from large errors due to inadequate convergence of the flow results.

It is noted that the current implementation is one of a few different approaches tested by the author in order to shorten the computational time. The current implementation and settings are satisfactory in terms of the compromise between the number of blade updates required and the total flow computation needed between blade modifications (since fewer blade updates imply a larger modification at each update, which therefore requires more flow iterations to compute for the larger changes in the corresponding flow-field). One option that requires fewer flow iterations is to change the blade shape with each flow iteration. This implementation is similar in principle to that adopted by Meauze (1982) who prescribed surface pressures on the blade whose surfaces are assumed to be movable but impermeable (see Chapter 2, section 2.3.2.1). Although this implementation is found to require the least number of flow iterations, the reduction in computational time in flow analysis is compromised by the need to update the blade and to re-mesh the computation domain after each analysis time-step. At present, further investigation is required to find the best implementation and settings for optimum computational efficiency.

In the context of the design, convergence is obtained when the blade changes become sufficiently small, indicating that the surface normal transpiring flow velocities have almost disappeared. The tolerance on the blade changes is set with respect to the axial chord length. A typical value of 0.1% axial chord length is generally found to be adequate for the current applications, which are presented in Chapter 7.
CHAPTER 6
APPLICATIONS OF DESIGN METHOD I

In this chapter, the developed design method based on the specification of the mass-averaged swirl velocity (designated Method I) is applied in two and three dimensions.

In both cases, the design procedure is first validated by applying it to reproduce an existing known geometry based on its mass-averaged swirl velocity, \( r\vec{V}_o \) (or \( \vec{V} \) in 2-D) distribution. Following which, several designs are carried out to demonstrate the methodology.

In two dimensions, the design validation is carried out with the UTRC rotor cascade. The method is then applied to the design of a generic compressor rotor cascade where some understanding in the use of the method for transonic design is gained.

The MEL transonic turbine nozzle described earlier in chapter 4 is used to validate the method in three dimensions. This is followed by a redesign attempt to modify the nozzle blade. Finally, the method is applied to modify NASA rotor 67.

An extended application of the present method in an optimization procedure based on the Simulated annealing algorithm has also been carried out. This is not described in the thesis but may be found in Tiow et al. (2000).
6.1 APPLICATIONS TO TWO-DIMENSIONAL CASCADES

The applications of the method formulated in two dimensions are presented here.

6.1.1 Design Validation: UTRC Rotor

In this section, the design method is validated using the UTRC rotor cascade encountered in Chapter 4. The mass-averaged tangential velocity, $\bar{V}_y$, in the bladed region (i.e. from the leading- to the trailing-edges of the profile) of the rotor is first obtained from the flow analysis described in Chapter 4, section 4.2. By specifying the $\bar{V}_y$ distribution as the design target and keeping the original tangential thickness fixed, the computation aims to reproduce the original blade profile starting from a different geometry, which is approximated using equation (5.29) given in section 5.1.6 of Chapter 5.

Figure 6.1: Validation of 2-D Method 1: Reproducing UTRC Rotor Profile

Figure 6.1 shows the result of the computation. The diagram on the left shows the progressive modifications made on the initial profile (denoted by the dashed line) before arriving at the final geometry.
The final shape (outlined by the black dots) and the target shape (outlined by the solid black line and shaded in grey) are compared in the diagram on the right, where the computed geometry is shown to be a close resemblance of the original shape.

The corresponding mass-averaged tangential velocity, $\bar{V}_y$ distributions are shown in figure 6.2 where the final and target distributions are represented by the black dots and solid line respectively.

![Figure 6.2: Validation of 2-D Method I: Target and Final Mass-Averaged Tangential Velocity Distributions](image)

A close match between the two distributions can again be observed. The agreement is within 0.5 %, which is the specified convergence criterion for the design procedure. From these results, the accuracy of the method is thus verified.

No major difficulty was encountered during the exercise. However, it is important to ensure that the same convergence criteria are applied in the flow analysis to obtain the target $\bar{V}_y$ distribution and in the design computation in order to attain the level of match presented here.
6.1.2 Generic Compressor Rotor Cascade

The proposed inverse procedure is demonstrated next in the design of a transonic compressor rotor cascade. The use of $rV_{\theta}$ in inverse methods (see sections 2.3.1.6 and 2.3.1.7, Chapter 2) has so far been reported mainly for the design of turbomachinery blades in subsonic (or at most, near sonic) flow. With the new method, the flow model is improved and the use of $rV_{\theta}$ can therefore be extended for the first time into the viscous, high transonic flow regime for blades with finite thickness.

Shock-free or supercritical designs have always been attractive in transonic turbomachines since the formation of a shock incurs losses and its interaction with the surface boundary layer can lead to flow separation. Following the work of Hobbs and Weingold (1984), it is now well understood that such designs may be achieved based on a prescribed surface pressure or velocity. According to their study, the primary requirements to achieve a supercritical compressor design are a smooth and continuous acceleration from the leading-edge to the peak suction Mach number, followed by a shock-free deceleration (see figure 6.3).

![Figure 6.3: Surface Mach Number Distributions of a Supercritical (Controlled Diffusion) Blade](Reproduced from Hobbs and Weingold, 1984)
This type of velocity or pressure gradient on the suction surface has been referred to as a "ski-jump" distribution (a variant of the "roof-top" distribution mentioned in chapter 1). The essential feature of this shape is to accelerate the surface flow on the suction side rapidly, achieving as high a blade loading as possible and allowing adequate length for the diffusion to take place gradually from the velocity peak to the trailing-edge.

Using the design philosophy of Hobbs and Weingold, supercritical designs have resulted from inverse methods prescribing surface pressure or velocity (for example, Schmidt and Berger, 1986 and Demeulenaere and Van den Braembussche, 1996).

Prior to the current study, it has not been clear whether the use of flow field-averaged quantities like the currently adopted \( r\vec{V}_o \) (\( \vec{V}_y \) in 2-D) could be applied effectively to such design problems that have traditionally been solved by considering the effect of shocks on the surface flow. Here, this possibility is demonstrated.

The Mach number distribution suggested by Hobbs and Weingold may be interpreted using terminology familiar in the current work. The smooth but high acceleration at the leading-edge to the suction peak position corresponds to a region of high loading. Similarly, the gradual flow deceleration from the peak suction position may be interpreted as a gentle decrease in the loading. Therefore, a \( \frac{d\vec{V}_y}{dx} \) distribution that would be equivalent to the suggested Mach number distribution is one that is smoothly varying and concentrated towards the front of the profile. A fore-loaded design is therefore the objective for this computation.

The test case chosen for the computation is a generic one whose flow velocity triangle is typical of compressor rotors described in open literature (e.g. Sanger, 1996). The inlet absolute flow angle is assumed to be 0.0 degree and the rotating cascade is simulated in the two-dimensional flow field with a fixed blade speed, \( U \) set at 240 m/s. Other input flow parameters are the inlet stagnation conditions which are set at \( T_0 = 300 \text{ K} \) and \( P_0 = 111896 \text{ Pa} \) and the exit static pressure which is fixed at 111958 Pa.
The pitch-to-axial chord ratio of the cascade is 0.83 and the tangential thickness of the rotor profile is chosen to be of the double circular arc (DCA) type (details are given in Appendix VIII) having a maximum of 9% axial chord located at the mid-axial chord position.

In terms of work requirement, the design is set to give a specific head rise, $\Psi' = \left( \frac{\bar{V}_{\text{exit}} - \bar{V}_{\text{entry}}}{U} \right)$ of 0.4 that corresponds to zero inlet swirl and an exit mass-averaged tangential velocity of about 96 m/s resulting from the compression.

The inverse design procedure must start with an initial geometry. In the absence of an existing profile with similar operating criteria that can be used to initiate the computation, the starting geometry is approximated. In this case, the initial camber geometry is predicted based on the specified mass-averaged tangential velocity, $\bar{V}_y$, using equation (5.29) given in section 5.1.6 of Chapter 5, and the target distribution of $\bar{V}_y$ which corresponds to a fore-loaded characteristic is generated using the three-segment method (see Chapter 5, section 5.1.5).

Figure 6.4 shows the Mach number contours and the corresponding distributions of mass-averaged tangential velocity and loading ($d\bar{V}_y / dx$) of the initial cascade.
Figure 6.4: Characteristics of the Initial Cascade

From the figure, the initial cascade geometry is clearly very poorly approximated as its loading is nowhere near the intended fore-loaded shape (see middle right hand plot in figure 6.4). It is noted that equation (5.29) has served well in numerous applications of the design method, but in cases with strong shock formation it becomes inadequate, as shown by the current situation.

Two main features of the initial cascade stand out. The first is the presence of a very strong shock standing normal across the passage and impinging on both surfaces of the cascade. The second is the strong diffusion along the suction surface, aft of the blade.
following the shock. Both features are clearly registered in the loading distribution; the shock formation as a sharp gradient between 25% and 65% axial chord position, and the diffusion aft of the blade as a steep unloading from the peak position. The loading distribution is therefore as good a gauge of such flow features as traditional properties like surface pressure or velocity, and is thus suitable as a design parameter in transonic design.

As the design computation progresses, its effects on the flow are traced. These are directly brought about by the changing geometry whose loading distribution is converging towards the specified shape as shown in figure 6.5, where the sequence of changes may be visualized.

![Figure 6.5: Mass-Averaged Loading Distributions: Initial, Intermediate and Final](image)

In relation to the flow field, the movement of the peak load position towards the front corresponds to the shifting of the passage shock in the same direction. This can be observed from the Mach number contour plots of figure 6.6.
In addition, it is noted that the decreasing peak load and gradient lead to a weakening shock whose effect is clearly illustrated by the diminishing shock patterns, also in figure 6.6.

**Figure 6.6: Relative Mach Number Contours: Initial, Intermediate and Final**

(Contour Interval: 0.015)

**Figure 6.7: Relative Surface Mach Number Distributions:**

*Initial, Intermediate and Final*
The gradual shaping of the "ski-jump" distribution through the design can be examined in figure 6.7. During the process, the peak Mach number level is reduced and its location is shifted forward. In the final design (blue lines), it may be observed that the Mach number distribution resembles that described by Hobbs and Weingold.

The geometrical shape of the initial, intermediate and final profiles are shown in figure 6.8.

![Figure 6.8: Comparison of Profile Geometries: Initial, Intermediate and Final](image)

The successful application of this design therefore conclusively shows the applicability of the current method and the effectiveness of the design parameter in transonic design. Details of the computation are given in the next section.

As mentioned at the start of this chapter, this method has been extended to form an automated optimization procedure based on the *Simulated annealing algorithm*. A continuation of the current work using the procedure was carried out in which the current "ski-jump" shape was scrutinized. An interesting result from the study shows that a more optimized supercritical design is achieved when an additional hump, which effectively gives a "double ski-jump" shape, is placed on the Mach number distribution. More details of the study can be referred to in Tiow et al. (2000).
6.1.2.1 Details of Computation

A total of 127 geometrical modifications are required in the current computation to complete the design process. The large amount of computation stems directly from the fact that the process has been initiated with a poor estimate of the initial shape where the starting discrepancy in \( \bar{V}_y \) is a considerable 85\% from the target value (see bottom right hand plot of figure 6.4).

A summary of the process is given in table 6.1.

<table>
<thead>
<tr>
<th>Blade Updates</th>
<th>Maximum Discrepancy From Target ( \bar{V}_y ) (%)</th>
<th>Number of Iterations Required in Flow Analysis</th>
<th>Computed Blade Changes (% axial chord)</th>
<th>Percentage of Flow Analysis Performed At this Stage</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>84.72</td>
<td>4320</td>
<td>9.220</td>
<td>6.70</td>
</tr>
<tr>
<td>1</td>
<td>74.64</td>
<td>4105</td>
<td>9.220</td>
<td>13.07</td>
</tr>
<tr>
<td>2</td>
<td>65.82</td>
<td>3976</td>
<td>6.945</td>
<td>19.23</td>
</tr>
<tr>
<td>3</td>
<td>56.06</td>
<td>3981</td>
<td>4.873</td>
<td>25.35</td>
</tr>
<tr>
<td>10</td>
<td>14.47</td>
<td>1781</td>
<td>1.225</td>
<td>58.97</td>
</tr>
<tr>
<td>20</td>
<td>7.17</td>
<td>885</td>
<td>0.488</td>
<td>75.65</td>
</tr>
<tr>
<td>30</td>
<td>4.97</td>
<td>384</td>
<td>0.241</td>
<td>82.90</td>
</tr>
<tr>
<td>40</td>
<td>3.36</td>
<td>194</td>
<td>0.127</td>
<td>86.72</td>
</tr>
<tr>
<td>60</td>
<td>2.11</td>
<td>85</td>
<td>0.087</td>
<td>90.855</td>
</tr>
<tr>
<td>80</td>
<td>1.68</td>
<td>85</td>
<td>0.052</td>
<td>93.59</td>
</tr>
<tr>
<td>120</td>
<td>1.14</td>
<td>85</td>
<td>0.025</td>
<td>99.04</td>
</tr>
<tr>
<td>127</td>
<td>&lt; 1.0</td>
<td>85</td>
<td>0.023</td>
<td>100.00</td>
</tr>
</tbody>
</table>

Table 6.1: Details of the Design Process
Due to the large starting discrepancy, the blade changes during the earlier stages of the computation are also the largest. The geometries between each design update differ substantially, as do their flow field solutions. As a result, more flow iterations are required as indicated in table 6.1.

A great proportion of the total computation is therefore involved in the first few design iterations; in fact, 60% (in terms of the total number of flow iterations required in the entire process) are made in the first 10 blade updates when the design convergence rate is the highest, reducing the initial discrepancy of 85% to 14.5% at this point.

As the computation proceeds, the convergence rate reduces since the driving term \((\bar{V}_x^* - \bar{V}_x^n)\) is now substantially lower (see equation 5.14 and the design convergence history in figure 6.9). Beyond the 40th geometrical update, the discrepancy between the current \(\bar{V}_x\) and the target specification is reduced to within 3.5% and the design process is now about 87% near completion.

In the final stages (beyond the 50th geometrical update), the blade modification becomes very small and the flow analysis only requires a small number of iterations to update the changes in the flow field. A minimum number of eighty-five flow iterations are performed when this situation emerges, to ensure that the small changes in the flow field are properly accounted for in the analysis. This is encountered from the 58th geometrical update onwards.

The overall convergence histories of the flow computation in the design process are shown in figure 6.10. In the plots, the spikes (better observed in the plot of \(\log (V_{rms\ error})\)) represent the sudden rise in the flow field error as the flow calculation is restarted after each blade update. There are therefore 127 spikes present in the histories.
Figure 6.9: Convergence Histories: Design

Figure 6.10: Convergence Histories: Overall Analysis
The computational requirement for the current process as performed on a single-processor DEC Alpha 433MHz workstation is given below in table 6.2.

<table>
<thead>
<tr>
<th></th>
<th>Strict Setting</th>
<th>Less Stringent Setting</th>
<th>Percentage Saving (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Mass Error: 0.5% V rms Error: 1.5 x 10^-3</td>
<td>Mass Error: 1.0% V rms Error: 5.0 x 10^-4</td>
<td></td>
</tr>
<tr>
<td>Number of Blade Updates</td>
<td>127</td>
<td>127</td>
<td>-</td>
</tr>
<tr>
<td>Total Flow Analysis</td>
<td>64473</td>
<td>38850</td>
<td>39.74</td>
</tr>
<tr>
<td>Iterations</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>C.P.U. Time</td>
<td>0:58:58.7 (hr:min:sec)</td>
<td>0:35:20.3 (hr:min:sec)</td>
<td>40.08</td>
</tr>
</tbody>
</table>

Table 6.2: Details of the Two Design Runs: Strict and Less Stringent Criteria on Flow Analysis

In the current computation, the flow analysis in design mode uses the same convergence criteria as that in the single flow analysis presented in Chapter 4. Arguably, such strict criteria are not required in the design mode since the sole concern here is the calculation of the blade shape and not the detailed flow results. Thus, although the calculations presented satisfied the strict criteria for the purposes of being consistent with Chapter 4 and testing the procedure stringently, the computational time may be improved by using less stringent settings, without compromising the final design.

One example is given in column 3 of table 6.2 where a repeated run of the current design is made using less-stringent criteria in the flow analysis (mass error: 1 % instead of 0.5% and V rms error: 5 x 10^-5 instead of 1.5 x 10^-5). By doing so, the computation is completed in about 60 % of the original CPU time and the design bears no significant differences from the original as shown in figure 6.11.
Figure 6.11: Comparison of Profile Geometries: Strict and Less Stringent Criteria on Flow Analysis
6.2 APPLICATIONS TO THREE-DIMENSIONAL BLADES

In this section, the three-dimensional version of the current method is applied.

As with the method in two-dimensions, the three-dimensional method is first validated and then applied to perform redesigns, which in this case involve the two three-dimensional test cases used to verify the flow solution, described in Chapter 4, sections 4.3 and 4.4.

6.2.1 Design Validation: MEL Turbine Nozzle

The validation of the three-dimensional method is performed using the annular turbine nozzle blade described earlier in Chapter 4, section 4.3.

![Figure 6.12: Validation of 3-D Method 1: Target Mass-Averaged Swirl Velocity Distribution](image-url)
The aim here is the same as for the validation carried out in two dimensions; that is, to verify the accuracy of the method by reproducing the original nozzle starting from a different shape, based on the specification of its mass-averaged swirl velocity distribution.

Figure 6.12 shows the specified \( r \bar{V}_\theta \) distribution at alternate span-wise grid positions across the meridional plane, as obtained from the calculation carried out in section 4.3.

The final result of the computation is shown in figure 6.13, where the blade section geometries at three blade heights -- near the hub, at mid-span and near the tip -- are shown. In each plot, the initial (dashed line), the target (solid line) and the final converged (solid dots) profiles are plotted together and the latter two can be compared.

From the plots, the computed blade geometry shows some discrepancy from the target shape. However, these differences are small and it may be observed from the figure that the computed blade has been generated to be a close match with the original nozzle blade.

The accuracy of the three-dimensional method is therefore sufficiently verified and, in the next section, is reapplied to redesign the blade.
Figure 6.13: Validation of 3-D Method I: Reproducing MEL Annular Nozzle
6.2.2 Redesign of the MEL Turbine Nozzle

In this section, the method is employed to redesign the nozzle blade. The sole objective of this is to demonstrate the methodology and hence, no exhaustive effort has been made to optimize performance. However, some forethought has been given to derive an improvement in aerodynamics.

The flow characteristics as revealed earlier by the flow computation in Chapter 4 are outlined here in order to recap the nature of the flow.

1) The flow is highly three-dimensional; the flow being transonic near the hub and subsonic near the tip.
2) The flow near the hub is highly transonic: Mach number peaks at about 1.8.
3) The diffusion aft of the blade at the hub is considerable.

Two main problems are commonly encountered in the design of transonic turbines; the first is unnecessarily high flow acceleration that may lead to shock formation, and the second is the strong diffusion that may take place on the suction surface when the flow decelerates from the peak velocity to equalize with that on the pressure side at the trailing-edge of the blade. In this instance, the annular nozzle does not suffer strong shock formation but the flow acceleration near the hub is high and as a result, the associated flow diffusion after the suction peak exceeds the generally accepted limit for diffusion factor, $D_F$ of 0.6 (see Chapter 4, section 4.3.3, figure 4.12, page 90).

The redesign therefore concentrates on reducing the diffusion following the suction peak aft of the blade.

The blade loading distribution, $\frac{\partial r \vec{V}_r}{\partial m}$, of the original blade is first studied; this is shown in figure 6.14 where the aft-loaded characteristics of the current nozzle blade may be observed.
As in the 2-D case presented in the last section, one can relate the current flow-field with its loading distribution. The positive gradient in the distribution along the stream-wise direction corresponds to the flow acceleration as it progresses through the narrowing cross-sectional area in the blade-to-blade passage, while the sudden drop following the acceleration represents the deceleration of the flow as it exits the cascade. Near the hub, the peak loading is the highest and the gradients are especially steep, reflecting the rapid development of the flow to supersonic state (see also Mach number contour plots in figure 4.11 of Chapter 4), followed by a sharp deceleration.

From the definition of the diffusion factor, $D_f = \frac{V_{\text{max}} - V_{\text{exit}}}{V_{\text{inlet}}}$, it is clear that by lowering the peak velocity, $V_{\text{max}}$, the diffusion on the suction side may be reduced. The redesign thus aims to limit the peak suction velocity and since the diffusion is highest in the transonic region near the hub, the modifications are made mainly in this region.
To achieve the objective, the current distribution of \( \frac{\partial r \hat{V}_\theta}{\partial m} \) at the hub is replaced with one that relocates some of the load from the aft to the front. This results in an improved loading curve with gentler gradients (before and after the peak position) and a lower peak load (see figure 6.16). The specification is generated using the three-segment algorithm described in section 5.1.5 of chapter 5. The differences between the original and the specified distributions of mass-averaged swirl and loading are shown in figures 6.15 and 6.16 respectively.

**Figure 6.15: Mass-Averaged Swirl Velocity Distributions at the Hub: Original and Redesign**

**Figure 6.16: Mass-Averaged Loading Distributions at the Hub: Original and Redesign**
At the tip, the loading distribution is also modified to give a lower peak load. Here, the change is made by defining a new \( r\bar{V}_\theta \) distribution based on the original by fitting a fifth-order polynomial through several selected control points. This and the corresponding \( \frac{\partial r\bar{V}_\theta}{\partial m} \) distribution are shown in figure 6.17.

![Figure 6.17: Modification of the Mass-Averaged Swirl Velocity Distributions at the Tip: Original and Redesign](image)

The specification across the whole of the meridional plane is then generated using linear interpolation between the distributions defined at the hub and tip. In this test case, the aft-loaded characteristic of the original blade is retained for the redesign. Figure 6.18 shows the resulting distributions of mass-averaged swirl, \( r\bar{V}_\theta \) and loading specification, \( \frac{\partial r\bar{V}_\theta}{\partial m} \) plotted at alternate span-wise grid points (these may be compared with the originals in figures 6.12 and 6.14 respectively).
Figure 6.18: Specification for the Redesign: Mass-Averaged Swirl Velocity and Corresponding Loading Distributions
The redesigned blade section at the hub is given in figure 6.19. It is found that as a gentler slope is prescribed for the loading distribution, the overall blade length increases. This is a direct consequence of redistributing some of the load (which is proportional to the blade turning) from the rear to the front region of the blade.

The direct implication of the design on the surface flow is shown in figure 6.20.
From the plots, it may also be observed that the suction peak in the new design is lowered substantially and, as intended, so is the diffusion* aft of the blade. As a result, the diffusion factor is successfully reduced to a more acceptable level, thereby satisfying the design objectives and confirming the possibility of controlling the diffusion via the loading distribution.

In a practical situation, the main motivation in weakening the diffusion aft of the blade is to reduce loss. The loss along the hub of the current design may be examined from the contours of entropy generation, \( \exp(-\Delta S/R) \) in figure 6.21; here, a value of 1.0 represents no thermodynamic irreversibility and a decreasing value indicates higher losses. From the contour plots, the apparent loss as a result of the strong diffusion in the original profile is noticeably reduced in the new design. An overall improvement of about 1% with respect to \( \exp(-\Delta S/R) \) is achieved in the new design over the original blade.

* The diffusion rate measured using exit condition (see footnote in page 90) is 0.067 for the redesign.
However, it is believed that there has been a compromise in the current level of improvement since a higher profile loss is incurred by the longer redesigned blade. This has been confirmed in a second redesign where the suction peak is lowered still further. In this case, the resulting blade is even longer and the loss involved, although lower than the original, is marginally higher than in the first redesign. An interesting study would be to examine the combined effect of the various causes of loss so that the optimum may be sought. To this end, the optimization procedure mentioned earlier is believed to be useful.

The three-dimensional pictorials of the original and the current redesigned blades are shown in figure 6.22.

Figure 6.22: Original and Redesigned Nozzle Blades
6.2.3 Redesign of the NASA Rotor 67

In the final example of this chapter, the method is used to redesign rotor 67. As revealed by the flow computation in Chapter 4, section 4.4, the flow in the rotor cascade is highly three-dimensional and the tip region is strongly transonic where a prominent passage shock is found.

The complex three-dimensionality of the shocked flow field is clearly represented in the \( \frac{\partial rV_\theta}{\partial m} \) distribution as shown in figure 6.23. Near the tip, the presence of the shock is distinctly registered as abrupt changes to the loading in the stream-wise direction. The shock in this region stretches across the passage and impinges on both sides of the blade. The suction and pressure legs of the shock show up as peaks on the distributions, which are indicated by the red dashed lines in the diagram.

![Figure 6.23: Mass-Averaged Loading Distributions of NASA Rotor 67: Strong Shock near the Tip Region](image)
In this design attempt, the objective is to demonstrate the methodology with an emphasis on modifying specific aerodynamic features, one of which is the shock structure.

The study of shock structure in such a flow field is very challenging since both its formation and its effects on the aerodynamics are highly complicated. In a three-dimensional case, visualizing the shock structure itself is a difficult task. Using a combination of experimental and computational methods, Wood et al. (1986) gave a good qualitative account of the shock surface in rotor 67. The reader is referred to their article for more details.

Here, the aim is to modify the shock structure by blade design, in particular using the mass-averaged loading distribution to shift the location and affect the intensity of the shock formation; an extension of the earlier study in 2-D (section 6.1.2). To bring about dramatic changes to the shock structure, a mass-averaged swirl velocity specification corresponding to a fore-loaded characteristic is specified along the tip (using the three-segment method) to shift the shock from the trailing-edge to the front end.

The second area of concern in this design is the high incidence in the hub region. Here, the original fore-loaded characteristic is kept but the variation is smoothed and the loading at the leading-edge is reduced (again using the three-segment method) so that the flow incidence in this region may be lowered. The mass-averaged swirl velocity distribution across the whole of the meridional plane is then, as before, generated by linearly interpolating the prescriptions along the hub and tip. The corresponding specification is shown in figure 6.24 and can be compared with the original in figure 6.23.
Fifty-one blade modifications are made in this computation. The process takes approximately 9 hours of C.P.U. time on a single-processor DEC Alpha 433MHz workstation to arrive at the final fore-loaded geometry, shown in figure 6.25.

In the following subsections, the main aerodynamic differences between the original and redesigned blades are described. The comparisons are made at three positions: near the hub, at mid-span and towards the tip. It is noted that although blade-to-blade treatment is, strictly speaking, inappropriate due to the three-dimensional nature of the shock structure, it is used in the discussion for conformity with current practice.
Figure 6.25: Original and Redesigned Blades
6.2.3.1 Near the Hub

In section 6.2.3, it is pointed out that the main concern of the design near the hub is the high flow incidence, which shows up distinctly in the loading distribution of figure 6.23 as large spikes near the leading-edge. The high incidence flow manifests itself as a region of recirculation as shown by the velocity vectors in figure 6.26. This is a clear sign of excessive incidence, which would adversely affect the stall margin of the machine.

![Figure 6.26: Relative Velocity Vectors at the Hub: Original Blade](image)

In the redesign, this situation is corrected by reducing the leading-edge loading so that a smooth entry flow may be recovered. Figure 6.27 shows the final results computed at 20% blade height.
Figure 6.27: Mass-Averaged Loading Distributions at 20% Blade Height: Original and Redesign

Three sections of change can be distinguished between the original and new loading distributions. The first is the intended decrease in load near the front, which is followed by an increase in the middle, and lastly, in the rear half of the axial chord length, a slight load reduction. As indicated in the figure, a correlation between the change in load and physical blade shape can be made. With increased loading, the blade camber becomes more pronounced, as shown by the middle section of the blade. At the front, with a lower leading-edge loading, the blade unloads and becomes more aligned with the inlet flow, thereby reducing the incidence angle. At the rear half, the slight decrease in the load is sufficient to produce a relatively straight back as compared to the original shape.

The design of the blade section near the endwalls encountered some difficulties (details are described later in section 6.2.3.5); nonetheless, the same trends are observed for the blade section at the hub, as shown in figure 6.28.
Figure 6.28: Comparison of Blade Profiles at the Hub: Original and Redesign

Figure 6.29 shows the entry flow to the new cascade at the hub. With the redesign, the recirculating flow previously observed near the leading-edge no longer exists at the entrance of the new blade.

Figure 6.29: Relative Velocity Vectors at the Hub: Redesigned Blade
6.2.3.2 Near the Tip

Near the tip, the blade camber is very slight, compared with the profile near the hub. In this region, the inlet Mach number is the highest (\(M=1.35\)) and most of the pressure rise here is caused by the shock. The shock wave is formed near the most convex part of the suction surface and is followed by strong diffusion near the trailing-edge.

In this case, the shock–boundary layer interaction at the tip is most intense and is enough to upset the boundary layer during diffusion and cause a region of separated flow (see figure 6.30).

![Figure 6.30: Relative Mach Number Contours and Velocity Vectors at the Tip: Original Blade](image)

In adopting a completely fore-loaded design, the intent is to shift the shock towards the front where the boundary layer is still thin and less likely to separate. However, as the inlet Mach number is already very high, this may cause stronger shock to develop at the front, incurring high losses. Ginder and Calvert (1987) encountered similar difficulties in the design of a civil aero-engine fan whose transonic aerodynamics are similar to that in rotor 67; they developed an optimization procedure to study and tackle the problem in a S1-S2 system.
In the current redesign, a high gradient is imposed on the prescribed blade loading distribution in the leading-edge region to accommodate the desired fore-loaded characteristic. However, the gradient has been limited so as to avoid a strong shock formation at the front.

The comparison of the redesigned and the original loading distribution near the tip along 80% blade height is given in figure 6.31. The steep slopes of the original loading distribution correspond approximately to the two ends of the passage shock, one on the pressure side and the other impinging on the suction surface of the blade. These are readily identified in the figure.

The effect of the redesign is also shown in the figure. From the Mach number contours of the new design, the passage shock has been relocated to the front as intended and without further aggravating its intensity. In fact, the shock is found to be milder especially on the suction surface where it is noticeably more diffused than that of the original.

These results reaffirm the effectiveness in using $\frac{\partial r \bar{V}_o}{\partial m}$ for transonic design; its peak position controls the shock location and the gradient directly determines its intensity of the formation. Here, the gentle slope of the middle section of the prescribed distribution is seen to be directly responsible for the weakened suction end of the shock formation.
Figure 6.31: Mass-Averaged Loading Distributions and Relative Mach Number

Contours at 80% Blade Height: Original and Redesigned
Careful examination of the new blade section reveals that the suction side is actually slightly de-cambered (or negatively cambered) (see figure 6.32) and it is this concave shaping of the suction surface that provides precompression of the flow, leading to a more diffused (i.e. weaker) shock formation.

Cumpsty (1989) explains that the gradual compression along the suction surface may be thought of in two ways,

1) The curvature towards the tangential produces compression waves which may coalesce into a series of weak oblique shocks; and
2) The cross-sectional area is decreased in the flow direction by this negative camber and this leads to a deceleration of the supersonic flow.
The use of precompression in transonic designs was first described by Prince (1980) where an example of a rotor blade with a very pronounced decambered section giving a distinctive "S" shape was designed and tested. The successful application of negative camber along the suction surface was also reported by Ginder and Calvert (1987) to yield an improvement in the aerodynamic performance of the civil aero engine fan design. The fact that supersonic flow expands along a convex surface and compresses along a concave one has also been recognized by Casey (1994) as a basis on which optimization procedures may be performed on transonic fans.

Along the tip section of the current design, an improvement is seen; the flow separation previously observed near the trailing-edge of the original blade has now disappeared (see velocity vectors in figure 6.33). This is a direct result of positioning the shock near the leading-edge since by doing so, not only is the shock moved to the region where the boundary layer is less susceptible to separation, there is also an adequate length for the boundary diffusion to take place gradually along the suction surface without separating the surface flow.
6.2.3.3 At the Mid-Span

Along the mid-span, the differences between the original mass-averaged loading distribution and that prescribed for the redesign are shown in figure 6.34.

Figure 6.34: Mass-Averaged Loading Distributions and Relative Mach Number Contours at 50% Blade Height: Original and Redesigned
Here, the fore-loaded design specification has a higher loading gradient at the front of the blade than in the original. This distribution is not directly controlled via the input parameter but is generated by linearly interpolating the specifications at the endwalls.

In the original design, a very weak oblique shock sits near the leading-edge of the cascade (see Mach number contour plot at the top of figure 6.34). With the redesign, this becomes more prominent as seen from the Mach number contours at the bottom of the figure. This is the direct consequence of the increased loading peak and gradient as indicated in figure 6.34.

Figure 6.35: Comparison of Blade Profiles at 50% Blade Height: Original and Redesigned

The blade section at this span position changes only slightly, but the decamber region is still noticeable, although much less prominent than that observed further towards the tip.
6.2.3.4 Overview of the Aerodynamic Changes

The changes in the flow field as a result of the redesign are summarized in this section. Figure 6.36 shows the Mach number contours on the surfaces of the blades where the changes in shock location and intensity across the meridional plane may be examined.

On the suction surface, the shifted and weakened shock formation at the tip is clearly represented by the contour pattern. The more intense shock formation near the midspan of the new design as pointed out earlier in section 6.2.3.3 may also be correlated with the contours shown in the bottom-left plot of the figure.

Figure 6.36: Meridional Views of the Surface Relative Mach Number Contours
(Contour Interval: 0.025)
On the whole, it may be observed that the supersonic region in the new blade has been narrowed. A three-dimensional picture of the changes may be formed in conjunction with the contour plot on the pressure surface of the blade as shown on the right of the figure. It may be observed that as the shock formation in the new design has been concentrated at the leading-edge (as shown in figure 6.33 and figure 6.34), a subsonic flow region now replaces the supersonic region that leads up to the shock in the original design. Here, the surface flow is found to be accelerating and then decelerating slightly in the stream-wise direction (this may also be noticed in figure 6.33).

This situation is more clearly represented in the surface static pressure distribution shown in the bottom-most plot of figure 6.37 where the slight dip and rise of the pressure level (which corresponds to the flow accelerating and then decelerating) on the pressure surface of the redesigned blade may be observed. This phenomenon arises as the fluid accelerates upon encountering the convex contouring of the pressure surface, which is positioned exactly where the surface on the reverse side is slightly decambered. A schematic diagram of the redesigned blade in the bottom-right figure has been exaggerated to illustrate this point with the red-dashed line indicating the approximate location of the precompression region.
Figure 6.37: Surface Static Pressure and Corresponding Surface Static Pressure Difference ($\Delta P$) Distributions: Original and Redesigned
A difficulty is encountered during the course of redesigning the fan blade, as mentioned in section 6.2.3.1. The ends of the blade (i.e. near the hub and tip) are observed to develop abnormally during the design process and this worsens as the design progresses towards convergence. The uncharacteristic development comes in the form of an abrupt "turning" of the blade near the hub and the tip.

As mentioned in section 5.1.5 of Chapter 5, the mass-averaged $r\tilde{V}_\theta$ is only specified along the hub and the tip and the distribution of $r\tilde{V}_\theta$ on the meridional plane is generated by linearly interpolating the two distributions. Linear variation of $r\tilde{V}_\theta$ is assumed across the entire span and does not take into consideration the influence of the viscosity on the flow in the regions near the end walls (i.e. the hub and tip). The simple radial profiling of the target $r\tilde{V}_\theta$ is found to differ significantly from the actual variation (see figure 6.38) and is believed to be the primary cause of the problem. The difference is obvious when the actual variation of the mass-averaged $r\tilde{V}_\theta$ (that is, in the presence of viscous influence) at the trailing-edge of the original fan is compared with the linearly interpolated distribution.

Figure 6.38: Radial Profiles of Mass-Averaged Swirl Velocity at the Trailing-Edge: Original, Specified and Final
It is then clear that the assumption of a linearly varying mass-averaged swirl velocity across the span is unsatisfactory close to the endwalls. Unfortunately, it is not clear what a universally realistic or satisfactory radial profile would be when prescribing the target specification. Initially, it was thought possible to adopt the original radial variation when generating the target specification near the end walls, but this restricts the design in two ways;

1) The assumption is only true when small blade changes are made. When substantial modifications are made, the influence of viscous effects near the end walls may change significantly with the blade changes. Irregular geometrical developments at the blade ends may still result, and

2) More importantly, by adopting the same variation, the design becomes questionable especially when existing undesirable phenomena due to the viscous influence near the end walls (for example, a separating boundary layer) are intended to be reduced. Adopting the original profile is thus not useful.

In the absence of large viscous effects (or when inviscid flow is assumed), linear radial profiling has produced reasonable geometrical shapes (e.g. in the redesigned nozzle blade). In this case, a compromise is made in which the fan blade sections near the end walls are not designed exactly to the specified $r\bar{V}_\theta$ distribution. Instead, they are defined by extrapolating the blade camber from the interior section (see figure 6.39).

![Extrapolated End](image_url)

**Figure 6.39: Modifying the Geometry Near the End-Casing**
It is accepted that this is restrictive and at best, only approximate for the design near the hub and tip. It is thus not ideal especially in this case when the geometrical details of the blade section in the transonic region are highly sensitive to shock formation and approximating the blade geometry near the tip region may not be satisfactory. Nonetheless, this does not completely negate the useful aspects of the current method as a design tool.

It does, however, suggest the need to define a useful design specification that would take into consideration the viscous influences when dealing with the regions near the end-walls; and also, the need for a more general method to generate the radial distribution of the design specification once the former is established.
6.2.3.6 Discussion: $r\bar{V}_\theta$ for Surface Flow Design

In figure 6.37 of section 6.2.3.4, the corresponding pressure difference (or $\Delta P$) distributions across the fan blades were also presented. As mentioned in Chapter 1, the design parameter of an inverse method may be a flow field averaged or surface flow parameter. The use of the latter is by far the more popular option since it most directly reflects the quality of the blade.

One reason for the successful application of $r\bar{V}_\theta$ in practical designs has been its relationship to $\Delta P$ and therefore the surface pressure distributions. As stated in Chapter 1, the theoretical relation for a thin blade in incompressible, irrotational flow, as given by Hawthorne (1984), is governed by equation (1.2),

$$\Delta P = P^+ - P^- = \frac{2\pi}{N} \left( \rho V_m \right)_{bl} \frac{\partial r\bar{V}_\theta}{\partial m}$$

(6.1)

Here, the relation of $\Delta P$ to $r\bar{V}_\theta$ is examined for the current case of a highly compressible and transonic flow field. Figure 6.40 compares the actual $\Delta P$ distribution of the original blade with the theory given by equation (6.1) but replacing $r\bar{V}_\theta$ with $r\bar{V}_\theta$ and using local blade-averaged density, $\rho_{bl} \left( = \frac{1}{2} \left( \rho^+ + \rho^- \right) \right)$ instead of the fixed flow density, $\rho$. In the region near the hub where the flow is mainly subsonic, the actual distribution is well predicted by the theory barring some discrepancy, which is thought to be mainly due to the fact that the effect of blade thickness is not accounted for in the equation.

Further from the hub, the theory continues to predict correctly the trend of the $\Delta P$ distribution. However, the discrepancy between the distributions increases as the flow becomes more transonic. At the tip, in the presence of the shock, the correlation deteriorates. The trend is still reasonably predicted but the intensity of the variation across the shock is poorly estimated. In the case of the redesigned blade, it may be observed that the correlation improves since the shock formation is less intense.
Figure 6.40: Comparison of Pressure Difference ($\Delta P$) Distributions: Actual and Theory
These observations therefore suggest that while the method has been demonstrated to yield the desired improvements in the current case, it must be noted that changes in the surface flow are not directly controlled, especially when the theoretical assumptions are not fulfilled. This is one shortcoming of the method.

In the next chapter, the method based on $\Delta P$ will be demonstrated.

6.3 CONCLUDING REMARKS

In this chapter, the first design method in two and three dimensions were tested and applied to several test cases. The following sub-sections summarize the content presented in this chapter.

6.3.1 Two-Dimensional Tests

In two dimensions, the method was tested in the design of a transonic compressor rotor cascade. In this early stage, some understanding of the use of mass-averaged swirl velocity, $\bar{V}_y$, and its derivative, $\frac{d\bar{V}_y}{dx}$, as the design parameter was drawn. The result shows that the peak load position coincided with the location of the passage shock in the cascade. Also, by adjusting the gradient of the loading distribution, the acceleration of the supersonic flow and thus the shock intensity may be controlled. The remainder of the chapter then concentrated mainly on ascertaining the relationship between shock formation and the variation of the mass-averaged blade loading distribution.
6.3.2 Three-Dimensional Tests

In three dimensions, the design method was used to redesign both the nozzle blade and the fan rotor blade, fulfilling the design intents of reducing flow diffusion (nozzle blade) and relocating the shock (fan rotor).

The fan redesign was carried out to further investigate the relationship between shock and the blade loading distribution. During the redesign, some difficulties were encountered in the computation of the blade geometry near the hub and tip. This suggests a limitation of the present method in linearly interpolating the target distributions defined on the hub and tip to generate the design specification across the meridional plane. The radial linear interpolation does not take into consideration the viscous influence of the flow near the end-walls and thus, produces irregular development of the blade near the hub and tip when the blade is modified to achieve the design specification in these regions. In the results presented, the geometries near the hub and tip were approximated by extrapolating the camber.

Despite the difficulties encountered, the design was able to shift and control the shock as intended, which was, in the case of the tip, from the rear to the leading-edge. Interestingly, the use of gentle gradient in the blade loading distribution to weaken the shock formation in the transonic region resulted in pre-compression regions, characterized by concave contouring of the suction surface of the redesigned blade. This coincides with previous published works where the decambering of the suction side has reportedly been used to control the shock formation in transonic fans and compressor rotors. In the published work, it emerged that the suitable shaping of the suction surface can weaken the shock formation. However, achieving a certain amount of pre-compression by altering the surface contour is difficult, because the shock formation is sensitive to the shaping of the blade surfaces (especially the suction side).

One of the more well-known and better described works in this area is that by Ginder and Calvert (1987) in their design of a civil aero fan. In their method, they adopted a qualitative design method in which the suction surface is defined using two arcs. The first is used to control incidence and the second to control pre-compression. Although the method has been highly successful, they noted that the method refers to a strictly
two-dimensional situation and no simple scheme relates the incidence in two dimensions to that in quasi- or full three-dimensional flow. In the design attempt using the present method, the problem of prescribing the desired amount of pre-compression is handled directly through the specification of the flow aerodynamics, i.e. in terms of the loading distribution.

The collective results obtained near the tip and along the mid-span in this redesign confirmed a direct positive relationship between the gradient of the mass-averaged blade loading distribution and the intensity of the shock.

The observations also demonstrated that some simple rules (for example, using a gentle gradient and avoiding abrupt changes) may be applied to the mass-averaged loading specification to control the development of shock in the design.

The aerodynamic improvements are, however, subject to the suitable prescription of the design parameter, which may be made based on a thorough understanding of the flow field. Optimization is seen as a viable option for this purpose.
CHAPTER 7
APPLICATIONS OF DESIGN METHOD II

This chapter presents the validation and application of the second design method in two and three dimensions. As with Method I, the design procedure is first validated by applying it to reproduce an existing blade before it is used to carry out other designs.

In two dimensions, the design method is validated using the 2-D UTRC stator cascade; this is shown in section 7.1.1. Following the validation, the capabilities of the 2-D procedure are demonstrated by producing several designs of a generic transonic turbine cascade, with emphasis on using the $\Delta P$ distribution to control the aerodynamics of the surface flow. In addition, the use of the Secant method to seek the correct $\Delta P$ distribution in order to obtain the required specific work (as described in Chapter 5, section 5.2.5.1) is also shown for one of the designs. This is presented in section 7.1.2.1.

The three-dimensional version of the method is first validated using the NASA rotor 67 fan blade. The method is then applied to redesign the blade, with the focus on improving areas of undesirable flow behaviour, namely the high incidence flow at the inlet near the hub and the strong shock formation near the tip. The overall performances of the new blades at the design and off-design conditions are then determined, and a qualitative improvement in terms of efficiency and working range is obtained.
7.1 APPLICATIONS TO TWO-DIMENSIONAL CASCADES

The validation and applications of the method in two dimensions are presented here.

7.1.1 Design Validation: UTRC Stator

In this section, the 2-D design method is validated with the UTRC stator cascade. The aim of the computation is to test the accuracy of the procedure in reproducing the known blade shape based on the specification of its $\Delta P$ and tangential thickness distributions. From the computation of the cascade flow field given in section 4.2 of Chapter 4, the distribution of blade surface static pressure difference ($\Delta P$) is obtained. This is imposed as the design specification and the design process is allowed to converge starting from a different initial blade shape.

Figure 7.1 shows the result of the computation. On the left, the progressive changes made in the initial cascade shape (dashed line) before arriving at the final shape are shown. The figure on the right shows the final outcome of the computation, where the profile finally computed (black dots) is compared with the target geometry given by the solid black line and shaded in grey.
When the design procedure converges, the final flow field is computed using the original solver code (i.e. "direct" mode calculation where the blade walls are solid). As mentioned in Chapter 5, section 5.2.1, the final "direct" mode flow computation is performed solely to check that the final $\Delta P$ distribution matches that specified in the design code. In this computation, the final pressure difference distribution (black dots) and the target distributions (solid line) are shown in figure 7.2.

![Figure 7.2: Validation of 2-D Method II: Target and Final Surface Static Pressure Difference Distributions](image)

Some differences between the two distributions can be observed. These are mainly due to the allowance given by the specified design convergence criterion (Chapter 5, section 5.2.7), implying that a small magnitude of transpiring flow may still exist along the surfaces of the blade profile in the final design iteration. To obtain a closer match in the $\Delta P$ values, the design may be allowed to converge further, thereby reducing the transpiring flow that still exists. In the current computation, however, the discrepancies between the final and target $\Delta P$ distributions are acceptably small and, as shown in figure 7.1, the original cascade shape is successfully replicated to a close tolerance. The accuracy of the present method based on the current design convergence criterion is therefore ascertained and the criterion is used for all the computations presented hereafter.
7.1.2 Generic Turbine Stator Cascade

The inverse design procedure is next applied in the design of a generic transonic turbine stator cascade. The generic cascade is invented to test the robustness of the procedure by generating several designs using different design specifications. In addition, the influence of the $\Delta P$ distributions on the design, and in particular on the surface static pressure distributions, is also examined.

The generic blade profiles are described with the British C4 series thickness (see Appendix VIII) with a maximum of 12.5 % axial chord and the cascade is set to have a pitch-to-chord ratio of 1.0. The transonic condition is given by choosing the ratio of the exit static pressure to the inlet stagnation pressure to be 0.55 and the inlet flow angle is fixed at 0.0 degree. In each design, the specified $\Delta P$ distribution is carefully adjusted by varying the area under the distribution (see section 5.2.5.1 of Chapter 5) such that the final cascade gives the same specific work (within 1%), which corresponds to zero mass-averaged tangential velocity at the inlet and 200 m/s at the outlet.

The computational domain used in all the designs consists of 21 nodes in the pitch-wise direction and 140 cells in the stream-wise direction, which include 51 points between the leading- and trailing-edges of the cascade.

Five different designs are carried out with this test case. The design specifications range from one with the peak surface static pressure difference positioned at the rear of the cascade (TD2D-1) to one whose peak is towards the front (TD2D-5). The design specification, $\Delta P$, is effectively the pressure loading of the cascade, and henceforth it is also termed the pressure loading distribution. Table 7.1 shows the pressure loading specifications used in the designs.
The first design, TD2D-1, is specified with an aft-loaded $\Delta P$ distribution which is generated using the three-segment method. In this case, the pressure loading is concentrated towards the trailing-edge of the blade section where its gradient is also the highest. The effect of this on the aerodynamics is to produce a very condensed local supersonic region at the rear of the resulting profile, which is clearly highlighted in the Mach number contours of figure 7.3 (a).

In Chapter 1, it was mentioned that one important feature of the present method is its ability to modify and control the surface pressure distribution along the blade and thus directly affect the surface flow. This is very important since the nature of the flow along the surfaces of a blade directly determines the quality of the blade. Detailed design rules based primarily on achieving certain surface Mach number or pressure distributions in different types of turbomachinery blades have therefore been formulated, for example, Starken (1989), Hobbs and Weingold (1984) (as outlined in Chapter 6 for supercritical...
compressor cascades) and Hourmouziadis (1989) for L.P. turbine blades. Prescribing the distribution of surface pressure (or Mach number) in inverse design methodologies has been problematic as explained in Chapters 1, 2 and 5. Here, the effectiveness of using $\Delta P$ to control the surface flow of the turbine cascade is demonstrated.

Figure 7.3 (a) shows that the variation of the specified $\Delta P$ distribution is reflected mainly as a variation in the suction surface pressure, while the distribution on the pressure side remains almost constant. In this case, therefore, it is observed that the specified aft-loaded $\Delta P$ distribution corresponds to the suction surface pressure decreasing to a minimum, whose position along the axial length is approximately where $\Delta P$ is the highest. Decreasing the pressure load from the peak position to zero at the trailing-edge corresponds to the surface flow diffusing sharply, as shown by the sudden increase in the static pressure towards the exit of the cascade.

From these observations, it is then clear how the surface pressure distribution (mainly the suction surface) may be controlled by prescribing $\Delta P$. For example, in design TD2D-2, it is shown that by specifying an evenly distributed $\Delta P$, a gentler sloping suction surface pressure distribution with a higher minimum point can be achieved (see figure 7.3(b)). The direct effect on the flow is a reduction of the surface flow acceleration and the peak velocity level on the suction side of the profile. These may be observed in the Mach number contours on the left of figure 7.3 (b), where a slightly less-pronounced shock formation and a lower peak Mach number level are shown.

In the next design, the objective of which is to reduce the flow diffusion on the suction surface towards the trailing-edge of the blade, the peak pressure load must be decreased gradually. This may be achieved by specifying a parabolic shape or a fore-loaded characteristic whose peak is located nearer the front, thus making a substantial portion of the blade length available for a gentle deceleration of the surface flow. These are shown in designs TD2D-3 and TD2D-4 in figures 7.3 (c) and (d) respectively, where a gentle rise in surface pressure (which corresponds to a mild flow diffusion) along the suction side aft of the profile is achieved.
It is also possible to limit the suction peak velocity and avoid strong flow diffusion in the same design. This is demonstrated in design TD2D-5 (figure 7.3 (e)) where the final suction pressure distribution is controlled such that it varies only slightly through most of the blade length (except at the front where most of the flow acceleration occurs). This is achieved by having a $\Delta P$ distribution that rises initially to a plateau followed by a gently descending slope at the rear half of the distribution. As shown on the left of figure 7.3 (e), the shock formation in the final cascade is weakened considerably and the peak Mach number is 1.195, compared to 1.413 in the first design, TD2D-1.

![Figure 7.3(a): Design 1: TD2D-1](image)

![Figure 7.3(b): Design 2: TD2D-2](image)
Figure 7.3(c): Design 3: TD2D-3

Figure 7.3(d): Design 4: TD2D-4
The results show conclusively that specifying $\Delta P$ as the design parameter is a feasible alternative to using the surface pressure distribution. They also show that with a relatively straightforward relationship between the surface pressure variations and the current design parameter, existing design experience based on achieving a certain type of surface pressure distribution can be adapted in the present method to give the desired designs.

In addition, the study shows that the present design method always produces blades that are structurally sound. This is one advantage over previous inverse methods based on the prescription of surface pressure.
7.1.2.1 Computation Using the Secant Loop

In all the examples presented in the last section, the target pressure loading distribution has to be adjusted to give the required specific work, quantified by the change in the mass-averaged swirl velocity from the inlet to the outlet of the cascade. This is one shortcoming of the current proposed method. As given in Chapter 5, section 5.2.5.1, the amount of change in the mass-averaged swirl velocity between the inlet and outlet can be varied by adjusting the area under the pressure loading distribution. This may be carried out either manually or automatically using the Secant loop.

The generic turbine design, TD2D-3 given earlier is generated using the automatic system. In this computation, the Secant method module is executed 20 times and a total of 42 geometrical modifications are made. The 20 pressure loading changes are controlled by the difference between the obtained net change in the mass-averaged tangential velocity in the cascade (i.e. $\bar{V}_{y_{\text{yx}}} - \bar{V}_{y_{\text{yl}}}$) and the target value which is 200 m/s for the current test case. Figure 7.4 shows the variation of $\bar{V}_{y_{\text{yx}}} - \bar{V}_{y_{\text{yl}}}$ in the computation while figure 7.5 shows the corresponding changes in the peak pressure load, $\Delta P_{\text{max}}$.

As indicated by the blue diamonds in the figures, when the value falls within a specified tolerance of the target value (1% in this case), the pressure loading distribution remains unchanged and the blade geometry is updated accordingly (red circles). Otherwise, the Secant loop is activated to modify the pressure loading distribution (denoted by the yellow diamonds) until the target $\bar{V}_{y_{\text{yx}}} - \bar{V}_{y_{\text{yl}}}$ is achieved. It may be observed from the figures that all 42 geometrical updates are performed when $\bar{V}_{y_{\text{yx}}} - \bar{V}_{y_{\text{yl}}}$ is within the specified band of acceptance.
Figure 7.4: Details of Computation: Variation of the Net Change between the Inlet and Outlet Mass-Averaged Tangential Velocities

Figure 7.5: Details of Computation: Variation of the Peak Pressure Loading
The geometrical changes undertaken in the design process are shown in figure 7.6, where the starting, final and an intermediate geometry obtained after six blade updates are given. The corresponding pressure loading distributions determined at these three stages of the design cycle are shown in figure 7.7.

![Initial, Intermediate and Final Geometries](image1)

**Figure 7.6: Initial, Intermediate and Final Geometries**

![Initial, Intermediate and Final Pressure Loading Distributions](image2)

**Figure 7.7: Initial, Intermediate and Final Pressure Loading Distributions**

The final cascade has an inlet mass-averaged tangential velocity of 0.0 m/s and produces an exit value of 200.3 m/s. This gives the net change in swirl to be within the 1% of the target value of 200.0 m/s; the successful working of the **Secant method** is thus demonstrated.
The total C.P.U. time for the computation is 46 minutes. This is about two and a half times that of a typical run without the Secant loop, but the advantage is that it does not require manual adjustment of the specified pressure loading and re-execution of the design procedure before the final design is obtained. The automated loop is found to be especially convenient and useful when applying the present method to a completely new design where the process must start from scratch. The strict tolerance of 1% used in the Secant loop is chosen for the purpose of testing the procedure stringently.
7.2 APPLICATIONS TO THREE-DIMENSIONAL BLADES

In this section, the three-dimensional version of the current method is tested and applied.

The 3-D method is first validated using the NASA rotor 67 fan rotor blade and then applied to modify the blade.

7.2.1 Design Validation: NASA Rotor 67

The computation of the flow field of the NASA designed fan blade was described earlier in section 4.4 of Chapter 4 and the computed pressure loading distribution across the entire meridional plane of the rotor is shown in figure 7.8.

For the validation, the distribution is imposed as the design specification in the design code and the objective, as in the validation carried out for the 2-D procedure in section 7.1.1, is to reproduce the original blade starting with a different geometry.
Figure 7.9 shows the final result of the computation where the blade shapes are compared at three blade heights; near the hub, at mid-span and near the tip. In each of the three plots, the initial (dashed line), the target (blue solid line) and the final profile (red solid dots) are shown. At all three span positions, it can be observed that the reproduced geometry matches the original shape closely.

Figure 7.9: Validation of 3-D Method II: Reproducing the NASA Rotor 67
7.2.2 Redesign of NASA Rotor 67

Two attempts are made to modify the original rotor 67 fan blade using the present procedure. A redesign of the fan blade was performed in section 6.2.3 of Chapter 6 using the first method, where the shock formation near the tip region was successfully shifted from the rear to the front of the blade by adjusting its \textit{mass-averaged loading} distribution. Here, improving the transonic aerodynamics is again one of the design objectives, but it is to be achieved by changing the pressure loading distribution.

The pressure loading distribution of the original blade and the relative Mach number contours at three blade heights are shown in figure 7.10. It is observed that there are similarities between the pressure loading and the mass-averaged loading distributions; the latter was shown in figure 6.23 of Chapter 6. In particular, it is noted that near the tip region where there is strong shock formation across the passage, the pressure loading, like the mass-averaged loading distribution, shows large variation. Also, in the region near the hub, the high flow incidence is reflected by the high leading-edge loading.

So far, the effects of adjusting the pressure loading have been seen in the 2-D designs of the generic turbine cascade. In this section, the capabilities of the 3-D version of the method are demonstrated by improving the design in the two areas where the flow aerodynamics have been less satisfactory, namely, the high flow incidence at the hub and the strong shock formation at the tip.

Two redesigns are performed. The first, \textit{FD3D-R1}, is specified to have a completely aft-loaded characteristic while the second, \textit{FD3D-R2}, is chosen to give a fore-loaded characteristic at the hub of the blade and a middle-loaded characteristic from the mid-span to the tip location. The following two sub-sections describe the two redesigns in more detail.
Pressure Loading:
Original Distribution

90% Blade

Shock Formation

70% Blade

High Incidence

30% Blade

Abrupt changes in pressure loading distribution near the tip correspond to the formation of shock

High pressure loading at the leading edge near the hub corresponds to high incidence flow

Figure 7.10: Original $\Delta P$ Distribution
(With Relative Mach Number Contours at Three Blade Heights)
7.2.2.1 Design FD3D-R1

In the first redesign, the pressure loading distribution is chosen to be aft-loaded. The required characteristics are specified at the hub (0% blade height) and at the tip (100% blade height) using the three-segment method. The distribution of $\Delta P$ across the whole meridional plane is then generated by linearly interpolating the values given at the hub and tip. The final design distribution plotted at alternate radial grid positions is shown in figure 7.11.

![Figure 7.11: Redesign Specification: Design FD2D-R1 (Aft-Loaded)](image)

It is noted that the redesign $\Delta P$ specification shown here has been checked to give a net change in the mass-averaged swirl velocity, i.e. $[r \bar{V}_{\theta sx} - r \bar{V}_{\theta tx}]$ that matches the value obtained by the original cascade to within 1%. This, as well as that used in the next design, i.e. FD3D-R2, is achieved by varying the total area under the distribution several times before the required $[r \bar{V}_{\theta tx} - r \bar{V}_{\theta sx}]$ is obtained by the final designs. The reader is referred to section 5.2.5 of Chapter 5 for more details.
Figure 7.12 shows the geometrical changes at the hub, mid-span and tip of the blade. In the figure, the corresponding surface static pressure and pressure loading distributions at these three blade heights are also compared to the original.

Near the hub region of the original blade, there is high static pressure loading at the leading-edge. As mentioned before, this represents a high incidence flow, which in section 6.2.3.1 of Chapter 6, was shown to cause a small region of flow re-circulation at the entry to the cascade.

In this redesign attempt, the static pressure loading at the leading-edge is reduced to zero. The reduction of the pressure loading at the front unloads the leading-edge of the blade and shifts the maximum blade turning towards the rear half of the axial chord, coinciding approximately with the position of the specified peak $\Delta P$. The resulting blade section shape is shown on the left of figure 7.12 (a).

The relation of blade turning to pressure loading is also seen at mid-span, where the final blade section has an obvious concentrated turn near its trailing-edge. This corresponds to the rear-loaded $\Delta P$ distribution specified at this span position (see figure 7.12 (b)).

At the tip, the profile of the original $\Delta P$ distribution can be related to the formation of the normal shock that stretches across the passage and impinges on both sides of the blade. It is interesting to note that the effects of the shock are registered separately; the first trough in $\Delta P$ corresponds to the shock on the pressure surface and the second, that on the suction side of the blade. The distinction therefore makes it possible to deal separately with the flow behaviour on either side of the blade. In this design, this is demonstrated by the specified aft-loaded $\Delta P$ distribution, which aims mainly to affect the flow behaviour on the pressure side of the blade while keeping the shock formation aft of the blade generally unchanged except shifting its position slightly. As a result, the main changes in the pressure distributions are on the pressure side, where the abrupt rise which existed previously is now replaced with a gentle slope (see figure 7.12 (c)). The effect of this on the shock formation is described later in section 7.2.2.3.2
Figure 7.12: Design FD3D-R1: Original and Redesigned Blade Geometries, Surface Pressure and Pressure Loading Distributions
7.2.2.2 Design FD3D-R2

The second redesign specifies the $\Delta P$ distribution to be fore-loaded at the hub and middle-loaded at the tip. As before, the specifications are defined using the three-segment method at the hub and tip and linearly interpolating them to give the final design distribution across the meridional plane. The final distribution plotted at alternate radial grid positions for this design is shown in figure 7.13.

![Pressure Loading: Middle Load Specification](image)

**Figure 7.13: Redesign Specification: Design FD2D-R2 (Fore/Middle-Loaded)**

As in the first design, the effect of the redesign on the blade geometry is first examined. This is shown in figure 7.14 (a)-(c) where the blade shapes and the corresponding changes in the surface static pressure and pressure loading distributions at three blade heights are compared.

The effect of reducing the high static pressure loading at the leading-edge of the blade near the hub was shown in the previous design. In that design, the leading-edge loading was reduced to zero, which resulted in major changes in the blade geometry, especially at the leading-edge where the blade profile is basically straight and aligned with the
inlet flow angle. Here, the leading-edge pressure loading at the hub is reduced but a finite value is kept. This results in the blade camber near the leading-edge being more prominent than that obtained in design FD3D-R1. Compared to the original blade, there is a change in the position where the blade turning is concentrated. The original design had a maximum camber at the rear of the blade while the new design has a relatively straight back with the maximum blade turning positioned nearer the front. A correlation between the blade shapes and the differences in their $\Delta P$ values may be observed in figure 7.14 (a).

At the mid-span, the pressure loading is smoothed and the differences between the redesigned and original distributions are relatively small compared to those at the hub. The blade modification at this section is therefore smaller as shown in figure 7.14 (b).

Along the tip, the original static pressure loading is changed drastically such that the pressure load in the rear half of the axial chord is redistributed to the front half. The redistribution gives an almost equal share of the total pressure loading between the two halves of the axial blade chord, and a smooth variation in $\Delta P$ is maintained from the leading- to the trailing-edge. The final pressure loading distribution and the corresponding changes in the surface static pressure distributions at this blade height are shown in the right hand plot of figure 7.14 (c).

As noted in the first design, the abrupt changes in the original $\Delta P$ distribution coincide with the locations where the shock impinges on the blade surfaces. The objective of having a smoothly varying $\Delta P$ distribution at the tip is to reduce the intensity of the shock formation. An early indication of having achieved this is observed from the surface pressure distributions, where the abrupt rises previously experienced by the original blade are now replaced by more gentle changes on both the suction and pressure sides of the blade. More details of the changes in the shock formation as a result of this design are described in the next section.
Figure 7.14: Design FD3D-R2: Original and Redesigned Blade Geometries, Surface Pressure and Pressure Loading Distributions
7.2.2.3 Overview of the Aerodynamics Changes

An overview of the aerodynamic differences between the original blade and the two new designs is given in this section; the focus being mainly on the inlet flow at the hub and the shock formation at the tip of blade.

7.2.2.3.1 High Flow Incidence at the Hub

The main undesirable flow phenomenon near the hub region of the original blade is the presence of flow re-circulation at the inlet. This is a direct consequence of having a highly loaded inlet region, which in terms of the flow aerodynamics, gives rise to a high incidence flow near the hub of the blade. The recirculating inlet flow is highlighted in the velocity vector plot of figure 7.16 (see top-most figure).

The redesigns thus aim to reduce the high positive incidence flow by alleviating some of the pressure loading in this region. This is performed by decreasing the pressure loading at the leading-edge (see figure 7.15).

![Figure 7.15: Pressure loading Distributions at the Hub: Original and Redesigns](image)

By doing so, both redesigns successfully reduce the flow incidence, thereby recovering a smooth flow at the inlet of the cascade. This is clearly shown in the velocity vector plots of figure 7.16 (see bottom two plots).
Figure 7.16: Relative Velocity Vectors at the Hub: Original Fan, Designs FD3D-R1 and FD3D-R2
7.2.2.3.2 Mach Number and Shock Formation

Near the blade tip, the flow aerodynamics present a different design difficulty. Here, the flow is predominantly transonic and the development of the supersonic region and shock formation in the flow passage are the main concerns of the redesigns.

The most serious aspect of shock formation in the cascade passage is its tendency to cause boundary layers to separate. At the tip of the original blade, the shock formation near the trailing-edge is at its worst and is sufficient to cause flow separation along the suction surface at the position just after the shock (see figures 7.17 and 7.18).

In the first redesign, the static pressure loading in the front half of the blade axial chord is increased to give a smoothed rear-loaded characteristic at the tip. As evident from the plot in figure 7.17, this gives rise mainly to a smoothed surface pressure distribution on the pressure side of the blade. In terms of shock formation, it is observed that the smoothed surface pressure distribution corresponds to the pressure leg of the shock formation being weakened considerably. This is represented by the diffused Mach number contours shown in the top middle plot of figure 7.17.

In design FD3D-R2, the pressure loading is specified to be middle-loaded at this blade height. With this distribution, the abrupt changes in the original $\Delta P$ distribution that correspond to the pressure and suction legs of the passage shock in the original cascade are replaced by a gentle variation (see bottom right-most plot in figure 7.17). In this case, the shock weakening takes place on both sides of the blade as may be observed in the Mach number contours of design FD3D-R2, given in the top right hand plot of figure 7.17.
Figure 7.17: Relative Mach Number Contours at the Tip: Original Fan, Designs FD3D-R1 and FD3D-R2
(Showing Relationships to the Static Pressure Distributions)
The velocity vectors in the trailing-edge region at the blade tip of the three blades are shown in figure 7.18.

**Figure 7.18: Relative Velocity Vectors near Trailing-Edge at the Tip:**
Original Fan, Designs FD3D-R1 and FD3D-R2

In design FD3D-R1, the flow near the trailing-edge no longer separates but a small region of flow diffusion can still be observed on the suction side (see middle plot of figure 7.18). This is caused by the suction leg of the shock formation, which is allowed to remain in this redesign (see section 7.2.2.1). In the second redesign, however, with the shock formation weakened at both surfaces of the blade, the shock-boundary layer interaction is reduced, thereby giving a smoothed surface flow in the final FD3D-R2 blade (see right-most plot of figure 7.18).

The presence of shock is a form of thermodynamic irreversibility leading to losses. The generation of entropy ($\Delta S$) in the flow field therefore provides a measure of loss in the flow field. The contours of the $\exp(-\Delta S/R)$ (where $R$ is the gas constant) at the blade tip section of the three blades are shown in figure 7.19, where decreasing values from 1.0 represent increasing levels of irreversibility.
From the contour plots, it may be observed that a substantial amount of entropy is generated across the normal shock in the original blade. In design FD3D-R1, there is still a distinct band of concentrated entropy generation across the position of the shock. However, the thermodynamic irreversibility is noticeably less, especially on the pressure side of the blade where the pressure leg of the passage shock has been weakened (see middle plot in figure 7.19). The right-most plot shows no obvious concentration of entropy generation as a result of the substantially weakened shock in design FD3D-R2.

The entropy generation contours at the mid-pitch position in the meridional view are shown in figure 7.20.
Here, it may again be observed that the entropy generation in design FD3D-R2 is the least intense among the three blades. In the next section, the operating characteristics of the new fans are compared with the original.
7.2.2.4 Performance Maps: Original and Redesigned Fans

Figures 7.21 and 7.22 show the adiabatic efficiencies and total pressure ratios of the original and redesigned blades plotted against the computed mass flow rate, which is normalized with respect to the choked mass flow rate of the original fan.

The maps compare the performance of the three blades. The numerical performance characteristics of the original blade and those of design FD3D-R2 are represented by the solid blue circle line and solid red square line respectively, whilst that of FD3D-R1 is denoted by the solid grey square line.

![Performance Maps](image)

**Figure 7.21: Adiabatic Efficiency: Original Fan, Designs FD3D-R1 and FD3D-R2**

(* Commercial Navier-Stokes Solver Code)

In terms of adiabatic efficiency, an improvement is achieved by design FD3D-2D, where a small increase (about 0.6 % at the design point) can be observed over most of the working range. On the other hand, the first redesign does not seem to yield any improvement.

In terms of total-to-total pressure ratios, the operating maps of the three blades are very similar, with the redesigns retaining the trend of increasing pressure ratios as mass flow rate is lowered. Comparing the characteristics of design FD3D-R2 with the original fan, there is only a marginal increase in the total pressure ratio delivered by the new fans.
The operating lines for design FD3D-R2 and the original fan relative to each other are checked using a commercial code — *TASCFlow (version 2.9)* — which solves the Navier-Stokes equations based on an implicit scheme and uses the standard $k-\varepsilon$ turbulence model to account for flow turbulence. The commercial code is used to provide three points of comparison on the maps for both the original and redesigned FD3D-R2 blades; these points are filled with yellow and represented by the circle and square legends respectively. As shown in the two figures, the relative position of the points computed by the commercial code for the two blades are in agreement with that indicated by the present code.

Besides an increased efficiency, the FD3D-R2 blade has an increased working range. In fact, the stall margins ($m_{\text{ratio,choke}}/m_{\text{ratio, stall}}$) of both the new blades are higher at 1.130 and 1.122 for designs FD3D-R1 and FD3D-R2 respectively, compared to 1.089 for the original blade. This is understood to be mainly due to the improvement made on the poor incidence flow near the hub of the original blade. As the high positive incidence near the leading-edge is reduced in the two redesigns, there is increased tolerance to the development of stall as the mass flow rate is reduced.
This explains the larger operating margin of design FD3D-R1 as compared to design FD3D-R2, with the former having an incidence angle of less than half that in the latter at the design point. Table 7.2 shows the incidence angle at the hub of the three blades at the design mass-flow rate.

<table>
<thead>
<tr>
<th>BLADE</th>
<th>ANGLE OF INCIDENCE (Degree)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Original Rotor 67</td>
<td>+37.32</td>
</tr>
<tr>
<td>Design FD3D-R1</td>
<td>+8.72</td>
</tr>
<tr>
<td>Design FD3D-R2</td>
<td>+19.13</td>
</tr>
</tbody>
</table>

Table 7.2: Angle of Incidence at Design Point

The entry flow velocity vectors near the hub of the original fan at its numerical stall point (i.e. the left-most solid blue points in figures 7.21 and 7.22) are shown in figure 7.23. A distinct region of high incidence flow with recirculating flow vectors can be observed. The recirculating flow region expands rapidly as the mass flow is reduced further. At approximately the same mass flow rate, the situations for the redesigned fans are somewhat better. The first redesigned fan, with zero leading-edge loading, maintains a smoothed entry flow (see bottom-left plot in the figure); this is also true for the second redesigned fan, but a closer examination reveals that the surface flow velocity vectors on the suction side of this blade are beginning to recirculate (see highlighted region in the bottom-right plot of figure 7.23). Nonetheless, the flow in this region is still noticeably better than that observed in the original cascade.
Figure 7.23: Relative Velocity Vectors at the Hub:
Original Fan, Designs FD3D-R1 and FD3D-R2
(Approximately at the Stalled Mass Flow of the Original Fan)
7.3 CONCLUDING REMARKS

The following sub-sections summarize the content presented in this chapter.

7.3.1 Two-Dimensional Tests

In two dimensions, the method was applied to produce several generic turbine stator cascades based on the specification of different types of pressure loading characteristics. The results showed conclusively that the use of pressure loading as the design parameter provides a good means of controlling the surface flow aerodynamics of the cascade. It was also demonstrated that it is relatively straightforward to specify $\Delta P$ in order to achieve a certain type of surface pressure distribution. Current design expertise in using the latter may thus be incorporated to improve blade performance.

The use of the Secant method to seek the required pressure loading in order to produce the specific work, showed the working of the automatic loop in conjunction with the present method. Although the coupled procedure was demonstrated with a parabolic design pressure loading distribution, the principle of the method can be applied to more general types of distribution, such as the three-segment distribution. More C.P.U. time is required when using the Secant loop, but this is compensated by the greater ease-of-use of the current design method, especially when a totally new design is needed.

7.3.2 Three-Dimensional Tests

In three dimensions, two redesigns were made of the NASA rotor 67 fan rotor blade. Design efforts concentrated on improving the poor incidence near the hub and strong shock formation near the tip. Although both redesigned blades are clearly not optimized, some improvements in the flow aerodynamics were made. The operating maps of both redesigns show that they have larger working ranges, and design FD3D-R2, which has a substantially reduced shock formation near its tip, gives an efficiency of about 0.6% higher than the original fan at the design point.
The qualitative improvement in the efficiency of design FD3D-R2 is confirmed by the computational results given by the commercial Navier-Stokes solver, TASCFlow (version 2.9).
CHAPTER 8
CONCLUSIONS AND SUGGESTIONS FOR FURTHER WORK

The traditional approach to blade design involves repeated tailoring of a blade shape and performing flow analysis to check the resulting blade performance. Such methods are inefficient, each iteration being guided solely by empirical rules and the designers' creativity. With increasing commercial pressure to produce high-performance custom blades in a shorter time, the development of more systematic computational design techniques is required.

In this thesis, the development of two new fully three-dimensional inverse methods for the design of turbomachinery blades is reported. The first method, designated Method I, uses the distribution of mass-averaged swirl velocity, \( r\tilde{V}_\theta \), as the design parameter; while the second method, Method II, uses the pressure difference, \( \Delta P \). Both methods were established to produce the required blade shape given their design specification. The following sections summarize the development of the two methods and outline the possible directions for further work.

8.1 THE FLOW SOLVER

Integral to both methods is a finite-volume, time-marching flow solver. The flow model is based on the unsteady, compressible Euler equations. To account for viscous effects, a body force model is included in the solution. Extensive validation of the solver was carried out using well-documented two- and three-dimensional test cases. In particular, a good correlation between the results obtained by the present solver code and the Navier-Stokes solver (N-S code, BTOB3D, by Dawes, 1987) was achieved for the modelling of the viscous transonic flow in the NASA rotor 67 fan blade.
8.2 METHOD I: BASED ON THE SPECIFICATION OF \( r\vec{V}_\theta \)

The motivation for the development of this method was the success of the design code, \( TurboDesign^1 \), of Zangeneh (1991) which is based on the specification of \( r\vec{V}_\theta \) and used by the industrial sponsor of the current project, \( Ebara Research Company Limited.\)

The code uses the potential flow model and has been versatile in designing centrifugal turbomachinery blades in the subsonic (or at most near sonic) flow regime. The new method based on compressible Euler flow with the inclusion of the viscous body force model extends the range of application to the viscous, high transonic flow regime. In addition, the theory has been formulated to allow thick blades to be designed.

The present methodology converges to give the final blade shape and the corresponding 3-D viscous flow field of the blade cascade. This is another difference from the previous implementation where the design code is based on the 3-D inviscid, potential flow model; in that method, if the viscous flow field is required, an external solver code, for example, the 3-D viscous codes of Denton (1990) or Dawes (1987), must be used.

The accuracy of the present method was verified by reproducing the 2-D UTRC rotor cascade and the 3-D MEL annular nozzle blade. Based on the specifications of their distributions of \( r\vec{V}_\theta \) and tangential thickness, but starting the computation with different camber shapes, the final geometries were successfully reproduced to a close match with the original shapes.

The application of the method to a 2-D generic test case showed that the mass-averaged swirl velocity (\( \vec{V}_\gamma \) in 2-D) may be used to control the transonic, shocked aerodynamics of a compressor rotor cascade. In particular, it was found that, by specifying the gradient of \( \vec{V}_\gamma \) (i.e. \( \frac{d\vec{V}_\gamma}{dx} \), the \textit{loading}), both the position and the intensity of the shock formation can be controlled effectively. It was also shown that the surface flow behaviour can be adjusted using the mass-averaged loading distribution. Although control of the surface.
flow is not directly applied, it is relatively straightforward to relate $\frac{d\vec{V}_y}{dx}$ to the surface flow characteristics. For example, in this case,

1) *The rate of increase in loading corresponds to the acceleration of the suction surface velocity of the blade;*

2) *Conversely, the rate of decrease in loading corresponds to diffusion of the surface flow; and*

3) *The position of the peak load position corresponds approximately to where the peak suction velocity is located.*

Using these relationships, the computation was able to produce a design that had strong flow acceleration near its leading-edge on the suction side of the profile, while shock-free, controlled flow diffusion was maintained at the rear of the profile. The resulting suction surface Mach number distribution had the characteristics of the well-known "ski-jump" shape advocated by Hobbs and Weingold (1984) for the design of supercritical compressor blade profiles.

In 3-D, the ability to control shocked aerodynamics using the loading distribution was confirmed. This was demonstrated with a redesign of the NASA rotor 67 fan blade. By adopting a fore-loaded distribution of the loading, $\frac{\partial r\vec{V}_p}{\partial m}$, the shock formation originally at the aft of the tip section was successfully shifted to the front. Also, by specifying a smooth loading distribution, the intensity of the shock formation was successfully moderated. Significantly, the resulting blade shape had a distinct precompression region (i.e. a decambered or concave section) on the suction surface. This is consistent with established design practice, where a decamber region is deliberately introduced in high-speed fan and compressor designs to limit the pre-shock Mach number level and the shock formation. The difference in this case is that the decamber region is a result of specifying a smooth loading distribution. It therefore does not require the designer to decide the correct amount of blade decamber in order to achieve a smooth flow field, thus making the design process more straightforward.
8.3 METHOD II: BASED ON THE SPECIFICATION OF $\Delta P$

In the review of the available inverse design methods, the use of surface pressure or Mach number distributions as the design parameter is by far the most popular. Specifying the surface distributions gives good control over the flow-field and allows improvements in the surface aerodynamics to be made directly. However, the methods based on the specification of the blade surface pressure or Mach number distributions are problematic. They generally face issues of having non-feasible blade shapes or convergence problems resulting from incompatible pressure distributions specified on the two surfaces of the blade and across the span. So far, there are no solutions to these fundamental problems. As a result, compromises are made. These may be to relax the design specification or to design only part of the blade shape.

For the second design method, the design parameter is the pressure difference between the surfaces of the blade. The quantity, $\Delta P$ is related to the surface pressures and is therefore a good alternative means to control the surface aerodynamics. In the present work, a new three-dimensional transpiration model was developed for the design code. The model allows the direct imposition of the $\Delta P$ values on the blade surfaces. If the target distribution is different from the one resulting from a direct calculation around the same geometry, the transpiring boundaries allow the convective fluxes to go through. The new blade shape is then calculated by aligning the blade surfaces with the transpiring velocity vectors. This method, like Method I, also allows the specification of a fixed tangential thickness distribution, thereby ensuring that the final blade shape is always structurally feasible.

The accuracy of the method was again satisfactorily verified by applying it to reproduce known blade shapes based on the specification of their $\Delta P$ distributions. In this case, the validations were performed with the 2-D UTRC stator cascade and the 3-D NASA rotor 67 fan blade.
In 2-D, the procedure was applied to produce five designs of a generic transonic turbine stator cascade, each having different $\Delta P$ distributions. It was thus demonstrated that by specifying the $\Delta P$ distribution, the surface static pressure distribution can be sufficiently controlled. In particular, it was shown that unnecessary flow acceleration and deceleration on the suction surface of the profile may be avoided by appropriately adjusting the $\Delta P$ distribution.

In the final application of the method in three dimensions, the NASA rotor 67 was redesigned. Two redesigns were presented and they concentrated on improving two main areas of the blade aerodynamics. One was the high incidence flow at the hub and the other was the strong shock formation at the tip section. The flow incidence at the hub was lowered by decreasing the $\Delta P$ value at the leading-edge, and the strong shock formation was weakened considerably by smoothing out the abrupt changes in the distribution of $\Delta P$. With the redesigns, larger working ranges were attained and for the second redesign, there was a 0.6 % improvement in the adiabatic efficiency.

8.4 CONTRIBUTION TO THE INVERSE DESIGN OF TURBOMACHINERY BLADES

In this section, some of the contributions of this study to the inverse design of turbomachinery blades are summarized.

The majority of the existing 3-D inverse methodologies use the surface pressure or Mach number distribution as design parameters. They are natural choices for controlling the surface aerodynamics of a blade, but the approach faces fundamental problems of being difficult both to use and to obtain feasible results.

Two alternative design parameters are adopted in the present study. They are namely, the mass-averaged swirl velocity, $\bar{rV_\theta}$ and the blade pressure difference, $\Delta P$. It has been established in the current study that these quantities are also effective inverse design parameters for producing blades with improved aerodynamic performance.
Earlier 3-D methods using swirl velocity are the Circulation methods, which are based on the potential flow model. They are limited to lower speed applications and cannot cope with shock formation. A more recent Euler-based circulation method (that of Dang and Isgro, 1994, 1995) is applicable to high-speed applications but the method is still restricted to zero thickness blades in pure inviscid flow.

The contribution of the current work to the development of methods based on $rV_\theta$ is that, for the first time, the application is extended to thick blades in viscous, transonic flow. It is also the only reported Euler-based method that uses the original flow analysis code in the design process (Tiow and Zangeneh, 2000). The implementation of this method is therefore simple and modular, and can be used with other existing Euler solver codes.

The method based on the specification of $\Delta P$ involves the development of a new transpiration model, as also reported by Dang (1995) and Ahmadi and Ghaly (1997) in 2-D. The present methodology is one of the first few 3-D inverse design methods that takes into account viscous flow effects for the design of high-speed turbomachinery blades with finite thickness. One difficulty of using this parameter is in maintaining the required specific work. In the current work, a possible means of overcoming this is established using an iterative outer loop based on the Secant method.

It is believed that the current work provides a strong foundation for future work in several key areas. The suggestions for possible further work are given in the next section.
8.5 SUGGESTIONS FOR FURTHER WORK

8.5.1 Enhancement of Convergence Acceleration and Improvement of the Grid System

One of the main components in the current methodologies is the Euler flow solver code. The present code is a basic finite-volume, time-marching solver, following the cell-vertex approach of Hall (1986) and using the multistage Runge-Kutta integrator with the artificial viscosity of Jameson et al. (1981). Local time-stepping and grid sequencing techniques have been implemented for convergence acceleration. However, this can be further enhanced by implementing the full multi-grid technique (Ni, 1981, Jameson, 1983). The current grid sequence method is meant to serve as the preparation for the full implementation of this technique.

The type of grid used in the current work is another area where improvements can be made. Although the H-grid is versatile and is able to represent different types of turbomachinery, it has shortcomings in modelling blunt leading- and trailing-edges. In these areas, C- or O- type grids will be better. However, because one of the considerations in the current development was the computational cost, H-grid topology was employed to alleviate the computational time required in regenerating the grid after each blade update. During the course of the work, there have been vast improvements in computer architecture and speed. It is thus foreseen that it will soon be computationally viable to employ a more sophisticated grid system with the current methods. Unstructured grid systems, particularly triangulation with adaptive meshing, will be useful for more accurate capturing of the shock formation. The method based on $\Delta P$ (Method II) is perhaps the preferred method to be employed with an unstructured grid system as it deals only with surface quantities; whereas Method I uses the mass-averaged swirl velocity in the pitch-wise direction, the computation of which in an unstructured grid system may not be easily performed.
8.5.2 Application to Radial Turbomachinery Blades with Splitters

The three-dimensional inverse methods described in this thesis were demonstrated for the design of axial flow blades. However, the methods are also applicable to the design of blades in radial configuration. The design of centrifugal compressor impeller or diffuser blades can therefore be performed.

![Figure 8.1: A Centrifugal Impeller with Splitter Blades](image)

In such cases, it is necessary to be able to design the splitter blades (figure 8.1) as well as the full blades. This extension can be carried out using two different distributions of $r\vec{V}_\theta$ or $\Delta P$; one for the full blade and one for the splitter. The specification must, however, take into account the spatial relationship between the two blades. This is because, if excessive loading is applied on the splitter blade, it may result in the blade overlapping with its neighbours. Constraints must therefore be imposed when specifying the design distributions.
8.5.3 Version for Incompressible Flow

For very low speed machines, the flow becomes almost incompressible and the low Mach number flow may impede the convergence of the time-marching solver. In such cases, especially for hydraulic applications, it is useful to implement a version of the current code that is based on incompressible flow. A step in this direction has been taken by the author in implementing design Method I with an incompressible solver code based on the artificial-compressibility method (Chorin, 1967). The current version for compressible flow is modified by changing the continuity equation to solve for pressure while keeping the density of the fluid constant. The formulation follows closely the work of Walker and Dawes (1990). Details of the preliminary work are given in Tiow (1999).

8.5.4 Application to Multi-stage Design

The formulation of the current methods is suited to the 3-D inverse design of blades in a multi-stage arrangement. The work will first involve an extension of the current codes to cope with additional blade row/s (see figure 8.2).

![Figure 8.2: Mesh Representation of a Two-Stage Turbine](Reproduced from Denton, 1990)

One of the main issues in multi-stage calculation is that the flow through consecutive blade rows in relative rotation is unsteady. The method of Denton (1990) removes the effects of unsteadiness by circumferentially averaging the flow at each axial station mid-
way between the blade rows. Another successful method of analyzing multistage flow is due to the work of Adamczky (1985) and Adamczky et al. (1990). This method, which is more complicated, involves decomposing the flow velocity into its passage time-averaged value, periodic and aperiodic fluctuating component and a term representing non-deterministic fluctuation, associated with turbulence and revolution-to-revolution variation.

The accuracy, robustness and feasibility of adopting either Denton or Adamczky's method for inverse design must first be studied. However, the development of inverse methods for multiple blade row design is very important. The complexity arising from interactions between the two blade rows could have significant consequences on the final blade shapes.

8.5.5 Inverse Optimization Method

Finally, the question of how to specify an optimum distribution of $r\tilde{V}_\theta$ or $\Delta P$ has to be answered. At the present stage, it is shown that by adopting some simple rules, the specification can be made to avoid detrimental flow behaviour, such as high flow diffusion, strong shock formation and so on. Although aerodynamic improvements can be made in these ways, it is not clear what the optimum distributions would be. Invariably, the optimum specification is case-specific and depends on the type of machine, the design condition and so on. In view of this, the development of inverse optimization procedures incorporating the current methods will be very useful.

As mentioned in Chapter 6, an early attempt in developing such a procedure has been made with Method I, using the Simulated annealing algorithm. The results have been encouraging and it will certainly be beneficial to further the study with Method II and in 3-D. It is envisaged that useful design databases can be set up in this way and, at a later stage, be incorporated with an Artificial neural-network to form a powerful inverse optimization, knowledge-based design system.
APPENDIX I:

EFFECT OF GRID SEQUENCING: SAMPLE TEST CASE (UTRC STATOR)

Solution of the hyperbolic unsteady Euler equation is limited by the need for numerous and small time steps. The speed of convergence depends linearly on the size of the time step. Larger time steps allow the wavelike mechanism, which eliminates transient disturbances, to propagate faster. The time step is limited by the CFL criterion and the grid size. The strategy behind mesh sequencing, as given by Laney (1998), is as follows;

1) Define a sequence of grids from coarse to fine where the fine grid corresponds to the next coarse grid but with double the mesh points in each of the two directions (for 2-D).
2) Begin calculation with initial conditions on the coarsest grid for steady-state solution; here, the transients travel very quickly to the boundaries and leave the domain.
3) The results on the coarse grid are interpolated to the next fine grid and again allowed to converge to steady state.
4) The process is repeated until the solution is found on the final and finest grid.

The sequencing strategy used in the present analysis code consists of up to three meshes: a coarse, a medium and a fine mesh.

A three-mesh sequence is demonstrated here for the analysis of the 2-D UTRC stator cascade whose details are given in Chapter 4, section 4.2. The convergence to steady state for this case is especially slow since the flow is almost incompressible, having a maximum Mach number of about 0.2.
The grid change sequence is as follows

1. **Refinement**
   - Coarse: 24 x 6
   - Medium: 47 x 11

2. **Final Fine**
   - Computational Domain: (stream x pitch) 93 x 21

**Figure I.1: Grid Change on UTRC Stator Cascade**

As given in table 3.1, a reduction factor of about 3 is achieved on the total time step and computational time for convergence to steady state solution.

| Level used | 2 |
| Machine C.P.U. | Dec Alpha 433Mhz (single) |
| Settings: | |
| Mesh | **Mass Error** | **V_{um}, Error** |
| Final Fine | 0.5% | 1.5 x 10^{-5} |

| No GS | 17135 | 2 GS | 5735 | Saving factor | 2.99 |
| No. of time steps | | | | |
| CPU time (hrs:min:sec) | 0:10:02 | 0:3:50 | 2.62 |

**Table I.1: Summary of Computation**
Figure 1.2: Convergence Histories on Analysis

It is noted that the current grid sequencing technique is a natural companion to the full multigrid and the full approximation scheme (FMG-FAS) (e.g. Jameson, 1983, Ni, 1981 and McCormick, 1987), which may be implemented at a later stage.

References:


APPENDIX II:

DIFFUSION FACTOR

In chapter 4, equation 4.1, the diffusion factor, $D_F$, is defined to be,

$$D_F = \frac{V_{\text{max}} - V_{\text{exit}}}{V_{\text{inlet}}} \quad (\text{II.1})$$

where $V_{\text{inlet}}$, $V_{\text{exit}}$ and $V_{\text{max}}$ denote the inlet, outlet and maximum velocity respectively.

This factor may also be approximated,

$$D_F \approx V_{\text{inlet}} + \frac{(\Delta V_\theta)_{\text{exit, inlet}}}{2} \cdot \frac{S}{c} - V_{\text{exit}}$$

$$= l - \frac{V_{\text{exit}}}{V_{\text{inlet}}} + \left| \frac{V_{\theta_{\text{exit}}} - V_{\theta_{\text{inlet}}}}{2} \cdot \frac{c}{S} \cdot V_{\text{inlet}} \right|$$

$$\quad (\text{II.2})$$

which is given by Lieblein (1965) who deduced the expression for the maximum velocity, $V_{\text{max}} \approx V_{\text{inlet}} + 0.5(\Delta V_\theta)_{\text{exit, inlet}} \cdot \frac{S}{c}$. Equation (II.2) may be an approximation but is usually the more convenient form to adopt since it does not require knowledge of the maximum velocity.

In either form, the diffusion factor is based on the establishment of the velocity gradient on the surfaces of the blade. It takes into account the effect of mean velocity change through the blade passage (i.e. the term, $\frac{V_{\text{exit}}}{V_{\text{inlet}}}$) and that due to the blade force required.
to provide the flow deflection (i.e. the term, $\frac{V_{\text{out}} - V_{\text{inlet}}}{2\cdot c_{\text{inlet}}}$ or $\frac{V_{\text{max}}}{V_{\text{inlet}}}$). The factor is therefore applicable for both compressor and turbine bladings. An alternative interpretation of the equation is given by Lakshminarayana (1996).

The relationship of the diffusion factor to the development of the surface boundary layer was established empirically from a large number of NACA tests (Lieblein and Johnson, 1961) carried out over a wide range of cascade geometries for a particular aerofoil section.

Figure II.1 shows an example of such a study for NACA 65-(A10) cascade blade. A general observation was that the boundary layer losses increase considerably beyond a diffusion factor of 0.6.

![Figure II.1: Variation of Friction Loss Versus Diffusion Factor](Reproduced from Lieblein, 1965)

Although the results are established in two-dimensional incompressible flows, the diffusion factor of 0.6 is commonly used as the guideline to assess the flow in working rotors and stators (Cohen et al., 1996).

Other forms of diffusion factor or ratios are also in use (for example, Japikse and Baines, 1994). The basic function of these factors in designs is the same. It is to give an
indication of the current state of diffusion relative to the permissible amount, which may be established from experience of blade testing.

In the current applications, no experimental blade testing is carried out. The diffusion factor is therefore used only as a qualitative measure where the factor of 0.6 is taken as the limit following Lieblein's results.

References


APPENDIX III:

GRID DEPENDENCY TEST FOR THE FLOW COMPUTATION OF NASA LEWIS ROTOR 67

A grid convergency test for the computations involving NASA rotor 67 is reported here. In the test, flow computations were made using five progressively refined grid definitions to determine grid independent solutions, and one grid setting is chosen for the final flow field computation and redesigns of the rotor blade.

The first grid is set A which has 14 grids in the span- and pitch-wise directions and 141 nodes in the stream-wise direction (14 x 141 x 14) giving a total of 27636 nodes. This grid set was chosen based on the work of Pierzga and Wood (1985) who employed a grid set consisting of 21021 grid nodes in the computational study of the rotor using Denton's (1982) Euler code with boundary layer coupling. Starting with this initial grid set, the number of nodes was increased in the span- and pitch-wise directions, giving rise to grid sets B (21 x 141 x 21), C (29 x 141 x 29), D (37 x 141 x 37) and the finest set, E consisting of 45 x 141 x 45 nodes. In the study, computations on all five grid sets are made at the same condition corresponding to the fan rotor operating at peak efficiency, and the same convergence criteria (see section 3.9, chapter 3) are applied in the iterative flow calculation.

At the chosen operating condition, the experimental value of the total pressure ratio between Aero-station 2 (see Chapter 4, section 4.4) and the inlet (i.e. $\frac{p_{\text{aero}}}{p_0}$) was given by Wood et al. (1985) to be 1.6470. Comparisons of this parameter with the computations are tabulated in Table III.1, with column 4 of the table indicating the percentage difference.
<table>
<thead>
<tr>
<th>Grid sizes</th>
<th>Total Grid Points</th>
<th>Total Pressure Ratio, $P_{\text{atm}0}/P_{\text{atm}}$</th>
<th>Percentage Difference from the Experimental value of 1.6470 (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$14 \times 71 \times 14$</td>
<td>27636</td>
<td>1.68468</td>
<td>2.288</td>
</tr>
<tr>
<td>$21 \times 141 \times 21$</td>
<td>62181</td>
<td>1.66749</td>
<td>1.244</td>
</tr>
<tr>
<td>$29 \times 141 \times 29$</td>
<td>118581</td>
<td>1.65398</td>
<td>0.423</td>
</tr>
<tr>
<td>$37 \times 141 \times 37$</td>
<td>193029</td>
<td>1.64880</td>
<td>0.109</td>
</tr>
<tr>
<td>$45 \times 141 \times 45$</td>
<td>285525</td>
<td>1.64875</td>
<td>0.106</td>
</tr>
</tbody>
</table>

**Table III.1: Grid Dependency Test: Single Parameter Check**

From the data, although it is observed that the computational results do not agree exactly with the experimental (due to experimental and computational uncertainties), improvements are made as more grid points are employed in the calculation. Doubling the number of nodes from the initial grid set A, the percentage discrepancy between the computational and experimental values is almost halved from 2.29% to 1.24% in grid set B. Increasing the number of nodes further in grid sets C, D and E, the computations match the experiment to within 0.5% difference.

The computation values obtained by the finer grid sets (C, D and E) are very closely matched, indicating that only slight improvement is made as the grid is refined from grid set C; between grid sets C and E, the improvement is about 0.3%. Between grid sets D and E, the 50% more grid points used in the latter (285525 grid points over 193029 in set D) resulted only in a marginal difference of less than 0.003%. Based on these results, grid independent solutions are assumed on these grid sets.

In addition to the overall total pressure ratio value, the radial profile of the ratio at the exit (Aero Station 2) is also compared and checked with the experiment. This is shown in figure III.1.
Satisfactory results are again seen on using grid sets C, D and E. From the current tests, it is clear that for single flow analysis, the most accurate result is achieved with the finest grid set, and for this reason, grid set E may be chosen. However, for the current inverse design procedure where repeated analysis of the flow field is required (redesigns of the rotor blade are made with both methods in Chapters 6 and 7), a compromise has to be made between computational time (and resources) and the grid-dependent accuracies. Grid set C is thus deemed a reasonable choice for the current redesigns, saving substantial computational time and storage over grid set D (which has 80% more grid points) with no great loss in the computational accuracy as shown in table III.1 and figure III.1. This grid set is therefore used in all the presented computations involving NASA rotor 67 (i.e. sections 4.4, 6.2.3, 7.2.1 and 7.2.2 of Chapters 4, 6 and 7 respectively).
References


APPENDIX IV:

TRANSFORMATION METRICS AND JACOBIAN

Consider the coordinate transformation from \((r, z) \rightarrow (\xi, \eta)\) such that,

\[
\begin{align*}
    r &= r(\xi, \eta) \\
    z &= z(\xi, \eta)
\end{align*}
\]

The terms involving the geometry of the grid, such as \(\frac{\partial \xi}{\partial r}, \frac{\partial \xi}{\partial z}, \frac{\partial \eta}{\partial r}, \frac{\partial \eta}{\partial z}\) are known as **Metrics**. In CFD applications, the transformation is given numerically, and hence the **metrics** are calculated as finite differences (Smith, 1985).

The metrics are partial derivatives expressed in terms of \(r\) and \(z\) as the independent variables. Consider a dependent variable in the governing flow equations, such as the velocity where \(V = V(r, z)\). The total differential of \(V\) is then given by,

\[
\begin{align*}
    dV &= \frac{\partial V}{\partial z} \, dz + \frac{\partial V}{\partial r} \, dr \\

\end{align*}
\]

therefore,

\[
\begin{align*}
    \frac{\partial V}{\partial \xi} &= \frac{\partial V}{\partial z} \frac{\partial z}{\partial \xi} + \frac{\partial V}{\partial r} \frac{\partial r}{\partial \xi} \\
    \frac{\partial V}{\partial \eta} &= \frac{\partial V}{\partial z} \frac{\partial z}{\partial \eta} + \frac{\partial V}{\partial r} \frac{\partial r}{\partial \eta}
\end{align*}
\]

Expressing equation (IV.2) in matrix form,

\[
\begin{pmatrix}
    \frac{\partial r}{\partial \xi} & \frac{\partial r}{\partial \eta} & \frac{\partial V}{\partial \xi} \\
    \frac{\partial z}{\partial \xi} & \frac{\partial z}{\partial \eta} & \frac{\partial V}{\partial \eta}
\end{pmatrix}
\begin{pmatrix}
    \frac{\partial \xi}{\partial r} \\
    \frac{\partial \xi}{\partial z} \\
    \frac{\partial \eta}{\partial r} \\
    \frac{\partial \eta}{\partial z}
\end{pmatrix}
= \begin{pmatrix}
    \frac{\partial V}{\partial r} \\
    \frac{\partial V}{\partial z}
\end{pmatrix}
\]
\( \partial V/\partial r \) and \( \partial V/\partial z \) are solved using Cramer's rule to yield,

\[
\begin{align*}
\frac{\partial V}{\partial r} &= \frac{\partial \xi}{\partial \eta} \frac{\partial \xi}{\partial \eta} - \frac{\partial \eta}{\partial \eta} \frac{\partial \xi}{\partial \eta} \\
&= \frac{1}{J} [V_{\xi} z - V_{\eta} z_{\xi}] \\
&= \text{(IV.3)}
\end{align*}
\]

\[
\begin{align*}
\frac{\partial V}{\partial z} &= \frac{\partial \xi}{\partial \eta} \frac{\partial \xi}{\partial \eta} - \frac{\partial \eta}{\partial \eta} \frac{\partial \xi}{\partial \eta} \\
&= \frac{1}{J} [V_{\eta} r - V_{\xi} r_{\eta}] \\
&= \text{(IV.4)}
\end{align*}
\]

Cramer's Rule:

Consider a set of equations,

\[
a_1 x + b_1 y = C_1 \quad \text{(IV.5a)}
\]

\[
a_2 x + b_2 y = C_2 \quad \text{(IV.5b)}
\]

Multiplying (a) Equation by \( b_2 \):

\[
a_1 b_2 x + b_1 b_2 y = C_1 b_2
\]

and (b) by \( b_1 \):

\[
a_2 b_1 x + b_1 b_1 y = C_2 b_1
\]

and subtracting, the two equations reduce to

\[
(a_1 b_2 - a_2 b_1) x = C_1 b_2 - C_2 b_1 \quad \text{(IV.6)}
\]
which then gives the values of $x$ and $y$ to be,

$$
x = \frac{C_1 b_2 - C_2 b_1}{a_1 b_2 - a_2 b_1} \quad \text{and} \quad y = \frac{C_2 a_1 - C_1 a_2}{a_1 b_2 - a_2 b_1}
$$

Representing the solutions as matrices,

\[
x = \begin{vmatrix} C_1 & b_1 \\ a_1 & b_1 \\ a_2 & b_2 \end{vmatrix} \quad \text{and} \quad y = \begin{vmatrix} a_1 & C_1 \\ a_2 & C_2 \\ a_1 & b_1 \end{vmatrix}
\]  \hspace{1cm} (IV.7)

The above is known as the Cramer's rule which is used in equations (IV.3) and (IV.4).

Reference

APPENDIX V:

DISCRETIZING THE BLADE UPDATE EQUATION: METHOD I

Discretization of the blade update algorithm (equation 5.27, Chapter 5, section 5.1.4.2) is carried out in the upstream and downstream directions from the fixed stacking location. Using subscript \( j \) to denote the bladed stream-wise location and subscript \( k \) to represent the grid location in the span-wise direction in the transformed plane, equation (5.27) is discretized using second order accurate finite central differencing.

Downstream Discretization with \( j = j_{\text{stack}} \) to \( j_{\text{RE-1}} \) & \( k = 1 \) to \( km \)

\[
A \theta^{n+1}(j+1,k) - A \theta^{n+1}(j,k) =
\left[ \frac{V_{\theta}^n(j+1,k)}{V_{\theta}^n(j+1,k)} \right] \left( A \theta^{n+1}(j+1,k+1) - A \theta^{n+1}(j+1,k-1) \right)
+ \frac{1}{2} \left[ \frac{V_{\theta}^n(j,k)}{V_{\theta}^n(j,k)} \right] \left( A \theta^{n+1}(j,k+1) - A \theta^{n+1}(j,k-1) \right)
\]

Rearranging and putting the term involving the camber modification at the location \( j = j+1 \) on the left hand side of the equation then yields,

\[
A \theta^{n+1}(j+1,k) + \frac{1}{4} \left[ \frac{V_{\theta}^n(j+1,k)}{V_{\theta}^n(j+1,k)} \right] \left( A \theta^{n+1}(j+1,k+1) - A \theta^{n+1}(j+1,k-1) \right) =
\]

\[
A \theta^{n+1}(j,k) - \frac{1}{4} \left[ \frac{V_{\theta}^n(j,k)}{V_{\theta}^n(j,k)} \right] \left( A \theta^{n+1}(j,k+1) - A \theta^{n+1}(j,k-1) \right)
+ \frac{1}{2} \left[ \frac{J(j+1,k)}{V_{\theta}^n(j+1,k)} \right] \left[ \frac{r \theta^*_{\theta}(j+1,k) - r \theta^*_{\theta}(j+1,k)}{r^2(j+1,k)} \right]
+ \frac{1}{2} \left[ \frac{J(j,k)}{V_{\theta}^n(j,k)} \right] \left[ \frac{r \theta^*_{\theta}(j,k) - r \theta^*_{\theta}(j,k)}{r^2(j,k)} \right]
\]

\( (V.1) \)
Upstream Discretization with \( j = j_{\text{stack}} \) down to \( j, k + 1 \) & \( k = 1 \) to \( km \):

\[
Af^{n+1}(j, k) - Af^{n+1}(j - 1, k) + \frac{1}{2} \left[ \frac{V_\xi^n(j, k)}{V_\eta^n(j, k)} \right] \left( Af^{n+1}(j - 1, k + 1) - Af^{n+1}(j - 1, k - 1) \right)
+ \frac{1}{2} \left[ \frac{V_\xi^n(j - 1, k)}{V_\eta^n(j - 1, k)} \right] \left( Af^{n+1}(j - 1, k + 1) - Af^{n+1}(j - 1, k - 1) \right)
= \frac{1}{2} \left[ \frac{J(j, k)}{V_\eta^n(j, k)} \cdot rV^*_\theta(j, k) - rV^n_\theta(j, k) \right] + \frac{J(j - 1, k)}{V_\eta^n(j - 1, k)} \cdot rV^*_\theta(j - 1, k) - rV^n_\theta(j - 1, k) \right]

\]

where \( Af^{n+1}(j_{\text{stack}}, k = 1 \rightarrow km) = 0 \).

Rearranging and now putting the term involving the camber modification at the location \( j = j - 1 \) on the l.h.s. of the equation then yields,

\[
- Af^{n+1}(j - 1, k) + \frac{1}{4} \left[ \frac{V_\xi^n(j - 1, k)}{V_\eta^n(j - 1, k)} \right] \left( Af^{n+1}(j - 1, k + 1) - Af^{n+1}(j - 1, k - 1) \right)
- Af^{n+1}(j, k) - \frac{1}{4} \left[ \frac{V_\xi^n(j, k)}{V_\eta^n(j, k)} \right] \left( Af^{n+1}(j, k + 1) - Af^{n+1}(j, k - 1) \right)
+ \frac{1}{2} \left[ \frac{J(j, k)}{V_\eta^n(j, k)} \cdot rV^*_\theta(j, k) - rV^n_\theta(j, k) \right] + \frac{J(j - 1, k)}{V_\eta^n(j - 1, k)} \cdot rV^*_\theta(j - 1, k) - rV^n_\theta(j - 1, k) \right]

\]

(V.2)

Equations (V.1) and (V.2) represent a tridiagonal system of equations. The Thomas Algorithm is used to decompose the equations and solve for the blade modifications at each quasi-orthogonal location.
APPENDIX VI:
DISCRETIZING THE BLADE UPDATE EQUATION: METHOD II

The final expression for the blade update algorithm for Method II is given by equation (5.45) of Chapter 5, section 5.2.4.1. Discretization of the partial differential equation is presented here. To compute for the new blade shape, the p.d.e. is integrated upstream and downstream from the fixed stacking point, \( j_{\text{stack}} \) using second order accurate central differencing. Using subscript \( j \) to denote the bladed stream-wise location and subscript \( k \) to represent the grid location in the span-wise direction in the transformed plane, the numerical integrations are carried out as follows,

Downstream Discretization with \( j = j_{\text{stack}} \) to \( j_{\text{TE}} - 1 \) & \( k = 1 \) to \( km \)

\[
\begin{align*}
& f^{n+1}(j+1,k) - f^{n+1}(j,k) \\
& = \frac{1}{2} \left[ \left( \frac{V_{\eta}^{n}(j+1,k)}{V_{n}(j+1,k)} \right) \left( f^{n+1}(j+1,k+1) - f^{n+1}(j+1,k-1) \right) \right] \\
& \quad + \frac{1}{2} \left[ \left( \frac{V_{\phi}^{n}(j,k)}{V_{n}(j,k)} \right) \left( f^{n+1}(j,k+1) - f^{n+1}(j,k-1) \right) \right] \\
& - \frac{1}{2} \left[ \frac{\Delta V_{\phi}^{n}(j+1,k)}{V_{n}(j+1,k)} \right] \left( t_{\phi}(j+1,k+1) - t_{\phi}(j+1,k-1) \right) \\
& - \frac{1}{4} \left[ \frac{\Delta V_{\phi}^{n}(j+1,k)}{V_{n}(j+1,k)} + \frac{\Delta V_{\eta}^{n}(j,k)}{V_{n}(j,k)} \right] \left( t_{\phi}(j+1,k) - t_{\phi}(j,k) \right)
\end{align*}
\]

where \( f^{n+1}(j_{\text{stack}}, k = 1 \rightarrow km) = f^{n}(j_{\text{stack}}, k = 1 \rightarrow km) \).
Rearranging and putting the term involving the new camber at the location \( j = j + 1 \) on the l.h.s. of the equation then yields,

\[
\begin{align*}
 f^{n+1}(j+1,k) \\
+ \frac{1}{4} \left( \frac{V_{\xi n}(j+1,k)}{V_{\eta n}(j+1,k)} \right) (f^{n+1}(j+1,k+1) - f^{n+1}(j+1,k-1)) \\
= f^{n+1}(j,k) - \frac{1}{4} \left( \frac{V_{\xi n}(j,k)}{V_{\eta n}(j,k)} \right) (f^{n+1}(j,k+1) - f^{n+1}(j,k-1)) \\
+ \frac{1}{2} \left[ \frac{J(j+1,k)}{V_{\eta n}(j+1,k)} \left( \frac{V_{\theta n}(j+1,k)}{r(j+1,k)} - \omega \right) + \frac{J(j,k)}{V_{\eta n}(j,k)} \left( \frac{V_{\theta n}(j,k)}{r(j,k)} - \omega \right) \right] \\
- \frac{1}{8} \left( \frac{\Delta V_{\eta n}(j+1,k)}{V_{\eta n}(j+1,k)} + \frac{\Delta V_{\eta n}(j,k)}{V_{\eta n}(j,k)} \right) (t_\theta(j+1,k) - t_\theta(j,k)) \\
- \frac{1}{16} \left[ \frac{\Delta V_{\xi n}(j+1,k)}{V_{\eta n}(j+1,k)} (t_\theta(j+1,k+1) - t_\theta(j+1,k-1) + \\
\frac{\Delta V_{\xi n}(j,k)}{V_{\eta n}(j,k)} (t_\theta(j,k+1) - t_\theta(j,k-1)) \right]
\end{align*}
\]

(VI.1)
Upstream Discretization with \( j = j_{\text{stack}} \) down to \( j_{\text{end}} + 1 \) & \( k = l \) to \( km \)

\[
\begin{align*}
& f^{n+1}(j, k) - f^{n+1}(j - 1, k) \\
& \quad + \frac{1}{2} \left[ \left( \frac{V^n_{\xi_0}(j, k)}{V^n_{\eta_0}(j, k)} \right) \left( f^{n+1}(j, k + 1) - f^{n+1}(j, k - 1) \right) \\
& \quad + \left( \frac{V^n_{\xi_0}(j - 1, k)}{V^n_{\eta_0}(j - 1, k)} \right) \left( f^{n+1}(j - 1, k + 1) - f^{n+1}(j - 1, k - 1) \right) \right] \\
& = \frac{1}{2} \left[ J(j, k) \left( \frac{V^n_{\xi_0}(j, k)}{r(j, k)} - \omega \right) + J(j - 1, k) \left( \frac{V^n_{\xi_0}(j - 1, k)}{r(j - 1, k)} - \omega \right) \\
& \quad + \frac{1}{2} \left[ \frac{\Delta V^n_{\xi}(j, k)}{V^n_{\eta}(j, k)} \left( t_\phi(j, k + 1) - t_\phi(j, k - 1) \right) \\
& \quad + \frac{\Delta V^n_{\xi}(j - 1, k)}{V^n_{\eta}(j - 1, k)} \left( t_\phi(j - 1, k + 1) - t_\phi(j - 1, k - 1) \right) \right] \\
& \quad - \frac{1}{4} \left[ \frac{1}{2} \left( \frac{\Delta V^n_{\xi}(j, k)}{V^n_{\eta}(j, k)} + \frac{\Delta V^n_{\xi}(j - 1, k)}{V^n_{\eta}(j - 1, k)} \right) \left( t_\phi(j, k) - t_\phi(j - 1, k) \right) \right]
\end{align*}
\]

where \( f^{n+1}(j_{\text{stack}}, k = 1 \rightarrow km) = f^n(j_{\text{stack}}, k = 1 \rightarrow km) \).
Again rearranging and putting the term involving the new camber at the location \( j = j - 1 \) on the l.h.s. of the equation then yields the expression giving the camber geometry downstream of the stacking line,

\[
f^{n+1}(j - 1, k) = \frac{1}{4} \left( \frac{V_{o_\theta}(j, k)}{V_{n_\theta}(j - 1, k)} \right) \left( f^{n+1}(j, k + 1) - f^{n+1}(j - 1, k - 1) \right)
\]

\[
= f^{n+1}(j, k) + \frac{1}{4} \left( \frac{V_{o_\theta}(j, k)}{V_{n_\theta}(j, k)} \right) \left( f^{n+1}(j, k + 1) - f^{n+1}(j, k - 1) \right) - \frac{1}{2} \left[ \frac{J(j, k)}{V_{n_\theta}(j, k)} - \frac{J(j - 1, k)}{V_{n_\theta}(j - 1, k)} \right]
\]

\[
+ \frac{1}{8} \left( \frac{\Delta V_{o_\theta}(j, k)}{V_{n_\theta}(j, k)} + \frac{\Delta V_{n_\theta}(j - 1, k)}{V_{n_\theta}(j - 1, k)} \right) \left( t_\theta(j, k) - t_\theta(j - 1, k) \right)
\]

\[
+ \frac{1}{16} \left( \frac{\Delta V_{o_\theta}(j, k)}{V_{n_\theta}(j, k)} - \frac{\Delta V_{n_\theta}(j - 1, k)}{V_{n_\theta}(j - 1, k)} \right) \left( t_\theta(j, k) - t_\theta(j - 1, k) \right)
\]

(VI.2)

To obtain the new blade camber, \( f^{n+1} \), the *Thomas Algorithm* is employed to solve the **tridiagonal** system of equations represented by equations (VI.1) and (VI.2).
APPENDIX VII:

THREE-SEGMENT METHOD

The three-segment method is used to define the target distributions for both design methods, that is,

1. **For Method I**: The target mass-averaged swirl velocity and loading; and
2. **For Method II**: The target surface static pressure difference distribution.

Originally introduced by Zangeneh *et al.* (1996) to specify the target swirl velocity (and the loading) distribution, as its name suggest, the method divides the distribution into three separate parts.

Parabolas are used to define the loading distribution for the first and last segments, and the middle segment is specified by a straight line with a given slope.

The following are necessary inputs to generate the target $r\bar{V}_\theta$ and $\partial r\bar{V}_\theta/\partial m$ distributions,

1. **Specify the position of NC and ND to separate the distribution into three parts.**
2. **Input mass-averaged swirl velocity at the leading-edge**, $r\bar{V}_{\theta le}$
   (or in 2-D, $\bar{V}_{y le}^*$).
3. **Input mass-averaged swirl velocity at the trailing-edge**, $r\bar{V}_{\theta te}$
   (or in 2-D, $\bar{V}_{y te}^*$).
4. **Specify the linear slope of the loading distribution**, $\partial r\bar{V}_\theta/\partial m$ in the middle segment.
5. **Specify the loading at the leading-edge**, $D_o$.

262
The respective equations for the loading, $\frac{\partial r \vec{V}_\theta}{\partial m}$ at each of the segments are as follows:

- **Parabolic 1st Segment:**
  \[ D_1 = A_1 m^2 + B_1 m + C_1 \]  \( \text{(VII.1)} \)

- **Middle Segment:**
  \[ D_2 = \text{Slope} m + C_2 \]  \( \text{(VII.2)} \)

- **Parabolic 3rd Segment:**
  \[ D_3 = A_3 m^2 + B_3 m + C_3 \]  \( \text{(VII.3)} \)

where $D_1$, $D_2$ and $D_3$ describe the $\frac{\partial r \vec{V}_\theta}{\partial m}$ distributions for the first, second and third segments respectively with $A_1$, $A_3$, $B_1$, $B_3$, $C_1$, $C_2$ and $C_3$ being real constants. Since $C_1 = D_1$ at $m = 0$ is given as an input (i.e. item 5 in the previous page) and $C_3$ can be determined knowing $D_3 = 0$ at the trailing-edge, of the seven constants, only five are unknown and they must be solved so that the equations are fully defined.

Three other equations are provided by integrating equations (VII.1), (VII.2) and (VII.3) to give the corresponding $r \vec{V}_\theta$ distributions.
Figure VII.2: Corresponding Swirl Velocity Distribution

They are as follows:

1st Segment:
\[ E_1 = \left( \frac{A_1}{3} \right) m^3 + \left( \frac{B_1}{2} \right) m^2 + C_1 m + F_1 \] (VII.4)

Middle Segment:
\[ E_2 = \left( \frac{\text{Slope}}{2} \right) m^2 + C_2 m + F_2 \] (VII.5)

3rd Segment:
\[ E_3 = \left( \frac{A_3}{3} \right) m^3 + \left( \frac{B_3}{2} \right) m^2 + C_3 m + F_3 \] (VII.6)

where \( F_1, F_2 \) and \( F_3 \) being real constants (\( F_1, F_2 \) being readily determined from the inputs listed in page 262). With the set of six equations, all the unknowns are then determined so that both the loading and swirl velocity distributions can be generated systematically.

For Method II, the required specification is the blade pressure difference distribution (or \( \Delta P \)). This is determined using the relation:

\[ \Delta P = K \frac{\partial r \tilde{V}_\theta}{\partial m} \] (VII.7)

where \( K \) is a real constant that is varied to satisfy the required specific work.

More details of the method of defining the target \( \Delta P \) have been given in Chapter 5, section 5.2.5.
Reference

APPENDIX VIII:

BLADE THICKNESS DISTRIBUTIONS

I) DOUBLE CIRCULAR ARC (DCA) THICKNESS

The symmetrical circular arc section of maximum thickness ratio, \( t/c = \tau \) is given by

\[
\frac{2y}{c} = \frac{l - \tau^2}{2\tau} + \frac{l}{2\tau} \sqrt{(l + \tau^2)^2 - 4\tau^2\bar{x}^2}
\]

where \( \bar{x} = \frac{2x}{c} - l \) and \( c \) is the axial chord.
II)  C4 THICKNESS

Figure VIII.2: C4 Thickness Profile

\[ y/c < 0.3 : \]
\[ y/c = A \left( 0.154920(x/c) - 0.061004(x/c)^2 - 0.283666(x/c)^3 + 0.332527(x/c)^4 \right) \]

\[ x/c \geq 0.3 : \]
\[ y/c = A \left( 0.040318 + 0.066147(x/c) - 0.118248(x/c)^2 + 0.017783(x/c)^3 \right) \]

where \( A \) is an arbitrary constant that controls the maximum blade thickness, \( t \)
\( c \) is the axial chord length
Maximum \( y = 0.005Ac \)

\( A = 1.0 \) gives a NACA C4 Thickness Distribution

Reference

REFERENCES


270


Dring, R.P., Joslyn, H.D., Hardin, L.W. and Wagner, J.H., "Research on Turbine Rotor- 
Stator Aerodynamic Interaction and Rotor Negative Stall", AFWAL-TR-81-2114, 
Components Branch (POTC) Turbine Engine Division (POT), Aero Propulsion 
Laboratory, Wright Aeronautical Laboratories, Wright-Patterson Air Force Base, Ohio, 
1981.


Dulikravich, G.S., "Aerodynamic Shape Design," AGARD Report 780, Inverse 

Dulikravich, G.S., "Aerodynamic Shape Design and Optimization: Status and Trends," 

Dulikravich, D.S. and Sobieczky, H., "Shockless Design and Analysis of Transonic 

Dunavant, J.C. and Erwin, J.R., "Investigation of a Related Series of Turbine-Blade 

18, No. 10, pp. 1153-1158, 1980.

Giles, M.B. and Drela, M., "Two-Dimensional Transonic Aerodynamic Design 

Ginder, R.B. and Calvert, W.J., "The Design of an Advanced Civil Fan Rotor," J. of 

paper 96-GT-158, 1996.


