1

PM-1 3½"x4" PHOTOGRAPHIC MICROCOPY TARGET
NBS 1010a ANSI/ISO #2 EQUIVALENT

Contributions to the Study of Turbomachinery Aerodynamics:

PART I: Design of a Fish-Tail Diffuser Test Section

***PART II: Computations of the Effects of AVDR on
Transonic Turbine Cascades***

By

**Michael F. Saroch
B.Eng. (Aerospace Engineering)**

A thesis submitted to
the Faculty of Graduate Studies and Research
in partial fulfilment of the requirements
for the degree of

**MASTER OF ENGINEERING
AEROSPACE ENGINEERING**

**OTTAWA-CARLETON INSTITUTE FOR MECHANICAL
AND AEROSPACE ENGINEERING**

**DEPARTMENT OF MECHANICAL AND AEROSPACE ENGINEERING
CARLETON UNIVERSITY
OTTAWA, CANADA
JANUARY, 1996.**

© 1996

Michael F. Saroch

National Library
of Canada

Acquisitions and
Bibliographic Services Branch

395 Wellington Street
Ottawa, Ontario
K1A 0N4

Bibliothèque nationale
du Canada

Direction des acquisitions et
des services bibliographiques

395, rue Wellington
Ottawa (Ontario)
K1A 0N4

Your file Votre référence

Our file Notre référence

The author has granted an irrevocable non-exclusive licence allowing the National Library of Canada to reproduce, loan, distribute or sell copies of his/her thesis by any means and in any form or format, making this thesis available to interested persons.

L'auteur a accordé une licence irrévocable et non exclusive permettant à la Bibliothèque nationale du Canada de reproduire, prêter, distribuer ou vendre des copies de sa thèse de quelque manière et sous quelque forme que ce soit pour mettre des exemplaires de cette thèse à la disposition des personnes intéressées.

The author retains ownership of the copyright in his/her thesis. Neither the thesis nor substantial extracts from it may be printed or otherwise reproduced without his/her permission.

L'auteur conserve la propriété du droit d'auteur qui protège sa thèse. Ni la thèse ni des extraits substantiels de celle-ci ne doivent être imprimés ou autrement reproduits sans son autorisation.

ISBN 0-612-13850-X

Canada

Name

MICHAEL T SARACH

Dissertation Abstracts International is arranged by broad, general subject categories. Please select the one subject which most nearly describes the content of your dissertation. Enter the corresponding four-digit code in the spaces provided.

AEROSPACE ENGINEERING

SUBJECT TERM

0538

U.M.I.

SUBJECT CODE

Subject Categories

THE HUMANITIES AND SOCIAL SCIENCES

COMMUNICATIONS AND THE ARTS
 Architecture 0729
 Art History 0377
 Cinema 0900
 Dance 0378
 Fine Arts 0357
 Information Science 0723
 Journalism 0391
 Library Science 0399
 Mass Communications 0708
 Music 0413
 Speech Communication 0459
 Theater 0465

EDUCATION
 General 0515
 Administration 0514
 Adult and Continuing 0516
 Agricultural 0517
 Art 0273
 Bilingual and Multicultural 0282
 Business 0688
 Community College 0275
 Curriculum and Instruction 0727
 Early Childhood 0518
 Elementary 0524
 Finance 0277
 Guidance and Counseling 0519
 Health 0680
 Higher 0745
 History of 0520
 Home Economics 0278
 Industrial 0521
 Language and Literature 0279
 Mathematics 0280
 Music 0522
 Philosophy of 0998
 Physical 0523

Psychology 0525
 Reading 0535
 Religious 0527
 Sciences 0714
 Secondary 0533
 Social Sciences 0534
 Sociology of 0340
 Special 0529
 Teacher Training 0530
 Technology 0710
 Tests and Measurements 0288
 Vocational 0747

LANGUAGE, LITERATURE AND LINGUISTICS

Language
 General 0679
 Ancient 0289
 Linguistics 0290
 Modern 0291
 Literature
 General 0401
 Classical 0294
 Comparative 0295
 Medieval 0297
 Modern 0298
 African 0316
 American 0591
 Asian 0305
 Canadian (English) 0352
 Canadian (French) 0355
 English 0593
 Germanic 0311
 Latin American 0312
 Middle Eastern 0315
 Romance 0313
 Slavic and East European 0314

PHILOSOPHY, RELIGION AND THEOLOGY

Philosophy 0422
 Religion
 General 0318
 Biblical Studies 0321
 Clergy 0319
 History of 0320
 Philosophy of 0322
 Theology 0469

SOCIAL SCIENCES

American Studies 0323
 Anthropology
 Archaeology 0324
 Cultural 0326
 Physical 0327
 Business Administration
 General 0310
 Accounting 0272
 Banking 0770
 Management 0454
 Marketing 0338
 Canadian Studies 0385
 Economics
 General 0501
 Agricultural 0503
 Commerce-Business 0505
 Finance 0508
 History 0509
 Labor 0510
 Theory 0511
 Folklore 0358
 Geography 0366
 Gerontology 0351
 History
 General 0578

Ancient 0579
 Medieval 0581
 Modern 0582
 Black 0328
 African 0331
 Asia, Australia and Oceania 0332
 Canadian 0334
 European 0335
 Latin American 0336
 Middle Eastern 0333
 United States 0337
 History of Science 0585
 Law 0398
 Political Science
 General 0615
 International Law and
 Relations 0616
 Public Administration 0617
 Recreation 0814
 Social Work 0452
 Sociology
 General 0626
 Criminology and Penology 0627
 Demography 0938
 Ethnic and Racial Studies 0631
 Individual and Family
 Studies 0628
 Industrial and Labor
 Relations 0629
 Public and Social Welfare 0630
 Social Structure and
 Development 0700
 Theory and Methods 0344
 Transportation 0709
 Urban and Regional Planning 0999
 Women's Studies 0453

THE SCIENCES AND ENGINEERING

BIOLOGICAL SCIENCES

Agriculture
 General 0473
 Agronomy 0285
 Animal Culture and
 Nutrition 0475
 Animal Pathology 0476
 Food Science and
 Technology 0359
 Forestry and Wildlife 0478
 Plant Culture 0479
 Plant Pathology 0480
 Plant Physiology 0817
 Range Management 0777
 Wood Technology 0746
 Biology
 General 0306
 Anatomy 0287
 Biostatistics 0308
 Botany 0309
 Cell 0379
 Ecology 0329
 Entomology 0353
 Genetics 0369
 Limnology 0793
 Microbiology 0410
 Molecular 0307
 Neuroscience 0317
 Oceanography 0416
 Physiology 0433
 Radiation 0821
 Veterinary Science 0778
 Zoology 0472
 Biophysics
 General 0786
 Medical 0760

EARTH SCIENCES

Biogeochemistry 0425
 Geochemistry 0996

Geodesy 0370
 Geology 0372
 Geophysics 0373
 Hydrology 0388
 Mineralogy 0411
 Paleobotany 0345
 Paleocology 0426
 Paleontology 0418
 Paleozoology 0985
 Polynology 0427
 Physical Geography 0368
 Physical Oceanography 0415

HEALTH AND ENVIRONMENTAL SCIENCES

Environmental Sciences 0768
 Health Sciences
 General 0566
 Audiology 0300
 Chemotherapy 0992
 Dentistry 0567
 Education 0350
 Hospital Management 0769
 Human Development 0758
 Immunology 0982
 Medicine and Surgery 0564
 Mental Health 0347
 Nursing 0569
 Nutrition 0570
 Obstetrics and Gynecology 0380
 Occupational Health and
 Therapy 0354
 Ophthalmology 0381
 Pathology 0571
 Pharmacology 0419
 Pharmacy 0572
 Physical Therapy 0382
 Public Health 0573
 Radiology 0574
 Recreation 0575

Speech Pathology 0460
 Toxicology 0383
 Home Economics 0386

PHYSICAL SCIENCES

Pure Sciences
 Chemistry
 General 0485
 Agricultural 0749
 Analytical 0486
 Biochemistry 0487
 Inorganic 0488
 Nuclear 0738
 Organic 0490
 Pharmaceutical 0491
 Physical 0494
 Polymer 0495
 Radiation 0754
 Mathematics 0405
 Physics
 General 0605
 Acoustics 0986
 Astronomy and
 Astrophysics 0606
 Atmospheric Science 0608
 Atomic 0748
 Electronics and Electricity 0607
 Elementary Particles and
 High Energy 0798
 Fluid and Plasma 0759
 Molecular 0609
 Nuclear 0610
 Optics 0752
 Radiation 0756
 Solid State 0611
 Statistics 0463
 Applied Sciences
 Applied Mechanics 0346
 Computer Science 0984

Engineering
 General 0537
 Aerospace 0538
 Agricultural 0539
 Automotive 0540
 Biomedical 0541
 Chemical 0542
 Civil 0543
 Electronics and Electrical 0544
 Heat and Thermodynamics 0348
 Hydraulic 0545
 Industrial 0546
 Marine 0547
 Materials Science 0794
 Mechanical 0548
 Metallurgy 0743
 Mining 0551
 Nuclear 0552
 Packaging 0549
 Petroleum 0765
 Sanitary and Municipal
 System Science 0790
 Geotechnology 0428
 Operations Research 0796
 Plastics Technology 0795
 Textile Technology 0994

PSYCHOLOGY

General 0621
 Behavioral 0384
 Clinical 0622
 Developmental 0620
 Experimental 0623
 Industrial 0624
 Personality 0625
 Physiological 0989
 Psychobiology 0349
 Psychometrics 0632
 Social 0451

The undersigned recommend to the Faculty of Graduate Studies
and Research acceptance of the thesis

**“Contributions to the Study of Turbomachinery Aerodynamics:
Part I: Design of a Fish-Tail Diffuser Test Section
Part II: Computations of the Effects of AVDR on
Transonic Turbine Cascades”**

submitted by Michael F. Saroch
in partial fulfilment of the requirements for
the degree of Master of Engineering

Thesis Co-Supervisor M.I. Yaras

Thesis Co-Supervisor S.A. Sjolander

Chair Department of Mechanical and Aerospace Engineering

Carleton University

Date

Jan 1996

Abstract

All gas turbine engines manufactured by Pratt and Whitney Canada today use fish-tail or pipe diffusers to guide and decelerate the flow from the exit of the centrifugal impeller to the inlet of the combustion chamber. Very little experimental analysis has been conducted on the flow through these diffusers. Part I of this thesis deals with the design and manufacture of a large scale fish-tail diffuser test section. The test section will be used for measuring the mean flow and turbulence quantities in the diffuser thus allowing a clearer understanding of the complex flow patterns present in such ducts.

* * * * *

There exists a lack of information in the literature regarding the effects of flow convergence and divergence on the performance of transonic turbine cascades. One measure of flow convergence and divergence is the axial velocity-density ratio (AVDR), or simply the axial velocity ratio (AVR) for incompressible flows. Rodger (1992) experimentally studied the effects of AVR on a cascade of turbine blades for low speed flow. Part II of this thesis deals with the computational investigation of the effects of AVDR on turbine blades operating at transonic conditions.

The computations were performed with a three-dimensional Navier-Stokes solver developed by Dawes (1988). An extensive survey of the code has been undertaken and a detailed description of the code is presented. The code was validated against Rodger's (1992) measurements, and although quantitative predictions were not very accurate the basic trends were predicted reasonably well.

Acknowledgments

I would like to thank my co-supervisor Professor S.A. Sjolander for his invaluable help during my research. His guidance during the writing of this thesis has been greatly appreciated. I would also like to thank my co-supervisor Professor M.I. Yaras for his support. His knowledge and guidance with regards to CFD have been invaluable.

The design and manufacture of the test section could not have been done without the help and cooperation of the Mechanical Engineering Technical Centre (ETC) and the Science Technology Centre (STC) with special thanks to J. Batky (ETC) and L. Boissonneault (STC).

I would also like to thank Pratt and Whitney Canada for their financial assistance.

Finally, I would like to thank my colleagues J. Verhiel, D. Jouini, and M. Jeffries for their assistance and support during my graduate studies.

Contents

Abstract	i
Acknowledgments	ii
List of Figures and Tables	viii
Nomenclature	xiv

PART I: Design of a Fish-Tail Diffuser Test Section

1 Introduction	1
2 Background	3
2.1 Diffusers for Centrifugal Compressors	4
2.1.1 The Vaneless Diffuser	4
2.1.2 The Vaned Diffuser	5
2.1.3 The Fish-Tail Diffuser	5
2.1.4 Impeller Discharge Flow	7
2.2 Straight Diffusers	9
2.2.1 General Description	9
2.2.2 Performance Parameters for Diffusers	10
2.2.3 Flow Regimes in Diffusers	11
2.2.4 The Effects of Inlet Conditions on Diffuser Performance	11
2.3 Constant Area Curved Ducts	12
2.4 Experimental Investigations of Diffusing Flow in Curved Ducts	13
3 Design of the Fish-Tail Diffuser Test Section	16
3.1 Geometry of the Fish-Tail Diffuser	17

3.2 Establishing Inlet Flow to the Diffuser	24
3.3 Wind Tunnel Contraction Design	26
3.4 The Fish-Tail Diffuser Test Section	29
3.4.1 Material	29
3.4.2 Machining of the Diffuser Test Section	30
3.4.3 Partitioning of the Diffuser Test Section	32
3.4.4 Tailpipe	35
3.4.5 Instrumentation	35
3.4.5.1 Seven-Hole Probe Measurements	35
3.4.5.2 Hotwire Measurements	37
3.4.5.3 Traversing Gear	40
3.4.6 Hardware	40
3.4.6.1 Mounting Plates	40
3.4.6.2 Flanges	42
3.4.6.3 Structural Support	42

4 Future Goals	45
-----------------------	-----------

PART II: Computations of the Effects of AVDR on Transonic Turbine Cascades

5 Introduction	47
6 Theory and Literature Review	51
6.1 Computational Fluid Dynamics for Internal Flows	51
6.1.1 Governing Equations for Fluid Flow	52
6.1.1.1 Full Navier-Stokes Equations	52
6.1.1.2 Thin Layer Navier-Stokes Equations	55
6.1.1.3 Parabolized Navier-Stokes Equations	57
6.1.2 Discretization of the Governing Equations	57
6.1.2.1 Finite-Difference Approach	58

6.1.2.2	Finite-Volume Approach	58
6.1.2.3	Finite-Element Approach	59
6.1.3	Discretization Errors and Numerical Dissipation	60
6.1.4	Methods of Solution	61
6.1.4.1	Explicit Methods	61
6.1.4.2	Implicit Methods	61
6.1.5	Turbulence Modelling	62
6.1.5.1	General Background	62
6.1.5.2	Mixing-Length Models	63
6.1.5.3	Eddy Viscosity Transport Models	65
6.1.5.4	Stress Transport Models	65
6.1.5.5	Turbulence Models for Internal Flows	66
6.1.6	Grid Generation	67
6.1.6.1	Structured Grids	68
6.1.6.2	Solution-Adaptive Unstructured Grids	68
6.2	Aerodynamics of Turbine Cascades	72
6.2.1	Loss Mechanisms in Turbine Cascades	72
6.2.1.1	Losses Due to Two-Dimensional Flow Phenomena	72
6.2.1.2	Losses Due to Three-Dimensional Flow Phenomena	74
6.2.2	The Effects of Axial Velocity Density Ratio on Cascade Flows	76
7	Computational Model	80
7.1	Three-Dimensional Navier-Stokes Solver (BTOB3D)	81
7.1.1	Governing Equations	81
7.1.1.1	Compressibility Considerations	83
7.1.2	Computational Mesh	83
7.1.3	Discretization of the Governing Equations	88
7.1.3.1	Finite-Volume Formulation	88
7.1.3.2	Solution Algorithm	89
7.1.3.3	Artificial Viscosity	90
7.1.3.4	Boundary Conditions	92
7.1.4	Multigrid Acceleration	95

7.1.5	The Baldwin-Lomax Turbulence Model	97
7.2	Simulation of Cascade Flow	100
7.2.1	Cascade Geometry	100
7.2.2	Mach Number Considerations	102
7.2.3	Convergence Criteria	104
7.2.4	Selection and Refinement of the Computational Mesh	106
7.2.4.1	Selection of the Computational Domain	106
7.2.4.2	Blade-to-Blade Mesh Refinement	107
7.2.4.3	Refinement of the Meridional Mesh	112
8	Presentation and Discussion of Results	119
8.1	The Effects of AVDR for Subsonic Conditions	120
8.1.1	Design Incidence	120
8.1.1.1	Profile Loss	120
8.1.1.2	Exit Flow Deviation	122
8.1.1.3	Base Pressure	123
8.1.1.4	Blade Static Pressure Distribution	124
8.1.2	Off-Design Incidence	128
8.1.2.1	Profile Loss	128
8.1.2.2	Exit Flow Deviation	130
8.1.2.3	Base Pressure	130
8.1.2.4	Blade Static Pressure Distribution	132
8.2	The Effects of AVDR for Transonic Conditions at Design and Off-Design Incidences	137
8.2.1	Profile Loss	138
8.2.2	Exit Flow Deviation	140
8.2.3	Base Pressure	140
8.2.4	Blade Static Pressure Distribution	143
8.3	Mach Number Effects	146
8.3.1	Profile Loss	146
8.3.2	Exit Flow Deviation	148

8.3.3 Base Pressure	148
8.3.4 Blade Static Pressure Distribution	150
9 Conclusions and Recommendations	151
9.1 Conclusions	151
9.2 Recommendations	153
Appendix 1A: Wind Tunnel Contraction Coordinates	155
Appendix 1B: Physical Properties	156
Appendix 2A: Description of BTOB3D	158
Appendix 2B: CC2 Blade Coordinates	170
Appendix 2C: Pre-Processor for BTOB3D	171
References	185

List of Figures and Tables

Chapter 2

Figure 2.1:	The geometry and notation of the vaned diffuser, according to Dean (1971), reproduced from Cumpsty (1989).	6
Figure 2.2:	The geometry and notation of the fish-tail diffuser installation.	6
Figure 2.3:	The idealized jet-wake model. (Reproduced from Dean and Senoo, 1960)	8
Figure 2.4:	The evolution of the flow pattern measured in a vaneless diffuser, according to Eckardt (1979), reproduced from Cumpsty (1989).	8
Figure 2.5:	Secondary flows in a curved duct. (Reproduced from Miller, 1978).	14

Chapter 3

Figure 3.1:	Flow path of a typical fish-tail diffuser. (Reproduced from Raw <i>et al.</i> , 1989)	18
Figure 3.2:	The installation of a fish-tail diffuser on the centrifugal impeller.	20
Figure 3.3:	Illustration of negative areas which arise for twisted cross sections.	21
Figure 3.4:	The area and cross-sectional shape distributions for the fish-tail diffuser.	22
Figure 3.5:	The geometry of the fish-tail diffuser flow path, including the tailpipe.	23
Figure 3.6:	Vortex generator system used to pre-condition the flow upstream of the diffuser inlet.	25
Figure 3.7:	Wall contour constructed of two matched cubic arcs.	28

Figure 3.8:	The geometry of the contraction flow path.	28
Figure 3.9:	Lamination of a typical half section prior to machining.	31
Figure 3.10:	Diffuser mesh. produced by CADKEY. used for CNC machining.	33
Figure 3.11:	Illustration of the fish-tail diffuser test section.	34
Figure 3.12:	Construction of the seven-hole probe.	36
Figure 3.13:	Near-wall measurements by the seven-hole probe.	36
Figure 3.14:	The seven-hole probe and traversing gear unit.	38
Figure 3.15:	Alignment apparatus for the seven-hole and hotwire probes.	39
Figure 3.16:	The geometry of a typical mounting plate.	41
Figure 3.17:	Installation of mounting plates on a typical lower half-section.	41
Figure 3.18:	'Male' and 'female' type flanges used for alignment and sealing between diffuser sections.	43

Chapter 6

Figure 6.1:	Example of an (a) O-type mesh (Reproduced from Raw <i>et al.</i> , 1989) and (b) C-type mesh (Reproduced from Arnore and Swanson, 1993).	69
Figure 6.2:	Example of an unstructured grid with solution-adaptive refinement in regions with high gradients. (Reproduced from Siden <i>et al.</i> , 1990)	71
Figure 6.3:	Dissipation coefficient for laminar and turbulent boundary layers. (Reproduced from Denton, 1993)	73
Figure 6.4:	Individual components of loss. $Re_{ex}=1,000,000$. (Reproduced from Mee <i>et al.</i> , 1992)	73
Figure 6.5:	The formation of secondary flows according to (a) Langston <i>et al.</i> (1977). and (b) Sharma and Butler (1987).	75
Figure 6.6:	Possible values of the axial velocity ratio (AVR). (Reproduced from Rodger, 1992)	77
Table 6.1:	Summary of Literature Survey from Rodger (1992).	79

Chapter 7

Figure 7.1:	The configuration and notation of the H -mesh.	84
Figure 7.2:	Illustration of the three families of surfaces found in the H -mesh.	85
Figure 7.3:	Illustration of the three different faces for the hexahedral cell.	87
Figure 7.4:	Calculation of area for the bladewise face of a hexahedral cell.	87
Figure 7.5:	Blade and cascade geometry.	101
Figure 7.6:	Mass-averaged total pressure loss vs. trailing-edge Mach number. (Reproduced from Isaacs, 1993).	103
Figure 7.7:	(a) RMS residue vs. time steps, (b) Maximum % error of mass flow through quasi-orthogonal planes, and (c) % Error of mass flow through selected quasi-orthogonal planes.	105
Figure 7.8:	Blade-to-blade mesh configuration: (a) Grid 1. (b) Grid 2. (c) Grid 3. and (d) Grid 4.	108
Figure 7.9:	Midspan mach number contours: (a) Grid 1. (b) Grid 2. (c) Grid 3. and (d) Grid 4.	109
Figure 7.10:	Pitchwise mass-averaged loss coefficient, Y' , at midspan for different mesh configurations.	111
Figure 7.11:	Computational mesh and midspan mach number contours: (a) Grid 4, and (b) Grid 5.	113
Figure 7.12:	Close-up view of (a) leading-edge and (b) trailing-edge region of grids 4 and 5.	114
Figure 7.13:	Meridional mesh with (a) endwall clustering, (b) no clustering, and (c) midspan clustering.	116
Figure 7.14:	Pitchwise mass-averaged loss coefficient, Y' , at $x/C_x=1.4$ for different meridional mesh configurations.	117

Chapter 8

Figure 8.1:	The effect of AVDR on midspan profile loss (Y'_{ms}) at design incidence for computational and experimental, subsonic results.	121
Figure 8.2:	The effect of AVDR on exit flow deviation (δ_{ms}) at design incidence for computational and experimental, subsonic results.	121

Figure 8.3:	Trailing-edge grid configuration.	124
Figure 8.4:	The effect of AVDR on the base pressure coefficient (C_{pb}) at design incidence for computational and experimental, subsonic results.	124
Figure 8.5:	Midspan static pressure distribution at design incidence for computational and experimental, subsonic results.	127
Figure 8.6:	The effect of AVDR on the midspan pressure distribution at design incidence for computational, subsonic results.	127
Figure 8.7:	The effect of AVDR on midspan profile loss (Y'_{ms}) at design and off-design incidences for computational, subsonic results.	129
Figure 8.8:	The effect of AVDR on midspan profile loss (Y'_{ms}) at design and off-design incidences for experimental, subsonic results.	129
Figure 8.9:	The effect of AVDR on exit flow deviation (δ_{ms}) at design and off-design incidences for computational, subsonic results.	131
Figure 8.10:	The effect of AVDR on exit flow deviation (δ_{ms}) at design and off-design incidences for experimental, subsonic results.	131
Figure 8.11:	The effect of AVDR on the base pressure coefficient (C_{pb}) at design and off-design incidences for computational, subsonic results.	133
Figure 8.12:	The effect of AVDR on the base pressure coefficient (C_{pb}) at design and off-design incidences for experimental, subsonic results.	133
Figure 8.13:	Midspan static pressure distribution at design and off-design incidences for computational, subsonic results.	134
Figure 8.14:	Midspan static pressure distribution, at design and off-design incidences for experimental, subsonic results.	134
Figure 8.15:	Midspan static pressure distribution at $+20^\circ$ incidence for computational and experimental, subsonic results.	135
Figure 8.16:	The effect of AVDR on the midspan pressure distribution at -10° incidence for computational, subsonic results.	136
Figure 8.17:	The effect of AVDR on the midspan pressure distribution at $+20^\circ$ incidence for computational, subsonic results.	136
Figure 8.18:	The effect of AVDR on midspan profile loss (Y'_{ms}) at design and off-design incidences for $M_{2s}=0.98$.	139

Figure 8.19:	The effect of AVDR on midspan profile loss (Y'_{ms}) at design and off-design incidences for $M_{2s}=1.15$.	139
Figure 8.20:	The effect of AVDR on exit flow deviation (δ_{ms}) at design and off-design incidences for $M_{2s}=0.98$.	141
Figure 8.21:	The effect of AVDR on exit flow deviation (δ_{ms}) at design and off-design incidences for $M_{2s}=1.15$.	141
Figure 8.22:	The effect of AVDR on the base pressure coefficient (C_{pb}) at design and off-design incidences for $M_{2s}=0.98$.	142
Figure 8.23:	The effect of AVDR on the base pressure coefficient (C_{pb}) at design and off-design incidences for $M_{2s}=1.15$.	142
Figure 8.24:	The effect of AVDR on the midspan static pressure distribution at design incidence for $M_{2s}=0.98$.	144
Figure 8.25:	The effect of AVDR on the midspan static pressure distribution at design incidence for $M_{2s}=1.15$.	144
Figure 8.26:	Midspan static pressure distribution at design and off-design incidences for $M_{2s}=0.98$.	145
Figure 8.27:	Midspan static pressure distribution at design and off-design incidences for $M_{2s}=1.15$.	145
Figure 8.28:	The effect of AVDR on midspan profile loss (Y'_{ms}) at design incidence for subsonic and transonic conditions.	147
Figure 8.29:	The effect of AVDR on the exit flow deviation (δ_{ms}) at design incidence for subsonic and transonic conditions.	147
Figure 8.30:	The effect of AVDR on the base pressure coefficient (C_{pb}) at design incidence for subsonic and transonic conditions.	149
Figure 8.31:	Midspan static pressure distributions at design incidence for subsonic and transonic conditions (AVDR=1).	149

Appendix 1A

Figure 1A.1:	Contraction Geometry	155
Table 1A.1:	Contraction Coordinates	155

Appendix 2B

Figure 2B.1: CC2 blade coordinates. 170

Table 2B.1: CC2 blade coordinates. 170

Appendix 2C

Figure 2C.1: Elliptic Clustering Strategy 172

Nomenclature

a_0	absolute stagnation speed of sound
A	area
C	blade chord length
C_x	axial chord length
C_f	skin-friction coefficient
C_p	pressure recovery coefficient (see definition in Section 2.2.2)
C_{pb}	base pressure coefficient (see definition in Section 8.1.1.3)
C_p	specific heat at constant pressure
C_v	specific heat at constant volume
D	adaptive artificial viscosity
e	specific internal energy
E_t	total energy
E, F, G	flux terms in Navier-Stokes equations (see definition in Section 6.1.1.1)
F_w	wall shear stress
h	static enthalpy
H	various fluxes in the Navier-Stokes equations in cylindrical coordinates
H	stagnation enthalpy
$\bar{H}_t$	effective total pressure loss (see Section 2.2.2)
I	rothalpy
I	identity matrix
I	linear interpolation function used in the multigrid method
k	thermal conductivity
k	kinetic energy of turbulence
l	turbulent mixing length
L	length
$\dot{m}$	mass flow rate = ρAV

M	Mach number = $V/\sqrt{\gamma RT}$
p	static pressure
P	forcing function used in multigrid method
P_b	base pressure
q	relative velocity
r	radius
r,θ,x	coordinates in the cylindrical coordinate system
R	gas constant
R	residual in the finite-volume formulation
Re	Reynolds number
s	blade span
S	body force terms in the Navier-Stokes equations
t	time
T	temperature
U	main stream velocity
U	conservation quantities in the Navier-Stokes equations (see definition in Section 6.1.1.1)
u,v,w	velocity components in x,y and z directions
V	volume
V	velocity
V_r,V_θ,V_x	velocity components in r,θ and x directions
W	weighting functions
x,y,z	coordinates in the cartesian system
x_m	location of the contraction match point
X	parameter describing contraction contour where: $X=x_m/L$
Y	profile loss coefficient (see definition in Section 8.1.1.1)
Y'	mass-averaged profile loss coefficient (see definition in Section 8.1.1.1)

Greek Letters

α	flow angle relative to the axial direction
β	blade metal angle relative to the axial direction
δ	flow deviation angle = $\alpha_{te}-\beta_{te}$

ρ	density
γ	ratio of specific heats, C_p/C_v
ϵ	turbulent dissipation rate
ϵ	free parameter used in the solution algorithm of Dawes code
ϵ	artificial viscosity coefficient
λ	second coefficient of viscosity
λ	spectral radius of the flux Jacobian
μ	dynamic viscosity
η	efficiency
π	shock sensor used in artificial viscosity terms
ξ, η, ζ	coordinate directions in computational space
τ	viscous stress
ω	vorticity
Ω	rotational speed

Subscripts

a	axial
B,S,Q	bladewise, streamwise, and quasi-orthogonal faces
i	isentropic conditions
i	inviscid component
i,j,k	indices of the computational mesh
le	leading-edge plane
ms	midspan
r, θ ,x	values in the three cylindrical coordinate directions
te	trailing-edge plane
T	turbulent
v	viscous component
x,y,z	values in the three cartesian coordinate directions
w	value at the wall
1	inlet plane
2	exit plane

Abbreviations

ASME	American Society of Mechanical Engineers
AVDR	axial velocity density ratio (see definition in Section 6.2.2)
AVR	axial velocity ratio (see definition in Section 6.2.2)
BTOB3D	version of the Dawes code used in the this study
CAD	computer aided design
CAM	computer aided manufacturing
CC2	Carleton cascade 2
CFD	computational fluid dynamics
CFL	Courant, Friedrichs, and Lewy number used as stability limit
CNC	computer numerically controlled
DNS	direct numerical simulation
ETC	Engineering Technical Centre - Mechanical
FAS	Full Approximation Scheme for multigrid
FNS	Full Navier-Stokes Equations
IGTI	International Gas Turbine Institute
LES	large eddy simulation
pvc	poly-vinyl chloride
P&WC	Pratt and Whitney Canada
PDE	partial differential equation
PNS	Parabolized Navier-Stokes Equations
PPNS	Partially-Parabolized Navier-Stokes Equations
RMS	root mean squared
STC	Science Technology Centre
TLNS	thin-layer Navier-Stokes equations

PART I:

**Design of a Fish-Tail
Diffuser Test Section**

Introduction

1

A diffuser is a device which decelerates the flow, converting as much of the dynamic pressure as possible to static pressure. Diffusers are used in a wide range of applications such as in engine intakes, wind tunnels, and gas turbine engines, to name a few. A wide range of diffuser geometries exist: straight (two dimensional), conical, and annular. More complicated geometries are often necessary for gas turbine applications. Curved diffusing ducts of arbitrary cross section are common in these applications.

Centrifugal compressors are commonly used in small gas turbines due to their high pressure ratios per stage and compact size. Diffusers are necessary to guide and decelerate the flow from the exit of the last centrifugal impeller to the inlet of the combustion chamber, with minimum loss and maximum flow uniformity. Traditionally, vaneless and vaned diffusers have been used; however, fish-tail diffusers have been preferred by Pratt and Whitney Canada (P&WC) due to their improved performance characteristics. Nevertheless, no comprehensive results have been published describing in detail the complex flow patterns present in these fish-tail diffusers. There exists a need for a detailed study of such flows.

The present work is concerned with the design and construction of a large scale fish-tail diffuser test section with the aim of providing physical insight into the complex flows occurring in these ducts. The work is related to fish-tail diffusers used on centrifugal compressors in small Pratt and Whitney Canada gas turbine engines.

Chapter two presents a background on present day diffuser technology and research. A brief description of the different types of diffusers used for centrifugal compressors, including the fish-tail diffuser is given. An overview of the terminology and the parameters used to measure the performance of straight-centerline diffusers is presented. A brief discussion of the flow through curved ducts with constant cross-sectional area is also given. Finally, a review of existing testing rigs for curved diffusing ducts is presented.

Chapter three presents a detailed description of the fish-tail diffuser test section, including the hardware and instrumentation used for taking measurements. A description of the inlet flow devices used to pre-condition the flow prior to entry to the diffuser is also given.

Chapter four discusses the future goals and intended direction of study with the fish-tail diffuser test section.

Background

2

Centrifugal compressors are commonly used in small gas turbine engines. Diffusers are necessary to guide and decelerate the flow leaving the last impeller to the entry of the combustion chamber. Section 2.1 briefly describes the different types of diffusers used for centrifugal compressors in today's engines. The main focus of the present study is the so called fish-tail diffuser.

Fish-tail diffusers exhibit high performance characteristics and are used in all engines manufactured by Pratt and Whitney Canada (P&WC) today. Similar curved diffusers are found in other applications such as engine intakes and exhaust ducts. Despite their wide use, limited work has been published on the flow through curved diffusers; particularly those where the radius of the bend is small compared to the inlet radius, as for fish-tail diffusers. However, a considerable amount of work has been done on straight diffusers and on flows through curved ducts without diffusion. A review of these more fundamental flows was deemed essential for guidance in the design and instrumentation of the present fish-tail diffuser test section. Section 2.2 presents some fundamental aspects of straight diffuser technology and Section 2.3 presents the same for constant-area curved ducts. Section 2.4 reviews the limited work which has been published regarding the flow through curved, diffusing ducts.

2.1 Diffusers for Centrifugal Compressors

Vaneless, vaned, and fish-tail diffusers are the typical choices for centrifugal compressors. The performance of these diffusers can vary greatly as can the cost. The following sections give a brief description of the three different diffuser types. A description of the flow exiting the centrifugal impeller is also presented since it has considerable influence on the performance of the diffusers.

2.1.1 The Vaneless Diffuser

Geometrically, the vaneless diffuser is the simplest and the most economical option consisting of an annular channel surrounding the impeller. Usually, the width of the channel is constant so that the diffuser is composed of two parallel, plane surfaces. The flow through vaneless diffusers has been extensively studied (eg. Dean and Senoo, 1960, Eckardt, 1975, and Inoue and Cumpsty, 1984) and is better understood than the flow through vaned or fish-tail diffusers. Vaneless diffusers are commonly used in turbocharger applications.

For the frictionless, incompressible case, with constant axial width, the streamlines in the diffuser form a logarithmic spiral from inlet to outlet on conservation of angular momentum grounds (Cumpsty, 1989). Even for the non-ideal case the flow will follow a long path. Typically, the tangential velocity entering the diffuser is about three times higher than the radial component of velocity. Consequently, it is the deceleration of the tangential component which gives most of the pressure rise. For the inviscid flow in a vaneless diffuser, the increase in radius is the only mechanism for decelerating the tangential velocity.

Besides low manufacturing costs, the vaneless diffuser benefits from a broader operating range than other diffusers, being less susceptible to stall instability (Cumpsty, 1989). For vaned diffuser and fish-tail diffuser configurations, there exists to a small extent a vaneless diffuser at the exit of the impeller. This is to avoid excessive pressure fluctuations, leading to high noise and fatigue on the diffuser inlet and impeller exit.

2.1.2 The Vaned Diffuser

Vaned diffusers are typically more efficient and give a higher pressure rise than vaneless diffusers. Most of today's high performance engines, utilizing centrifugal compressors (excluding P&WC engines), use vaned diffusers.

The most important effect of the vanes is to deflect the streamlines to a more radial path than that of a logarithmic spiral. Thus, the fluid particles travel along a shorter path to reach the same radius.

A wide variety of vaned-diffuser geometries are available. Figure 2.1 illustrates a wedge-type vaned diffuser; other designs use cambered vanes or aerofoil sections instead of wedges. According to Cumpsty (1989), there is little evidence that the vane geometry has any great significance on the flow. More important parameters, influencing performance and stability, are the number of vanes and the throat area.

2.1.3 The Fish-Tail Diffuser

The fish-tail diffuser was developed by J.C. Vrana and variations of it are used in all current Pratt and Whitney Canada engines (Sjolander and Saravanamuttoo, 1989). As for the vaned diffuser, the pipes deflect the streamlines to a more radial path than that of a logarithmic spiral. The pipes also turn the flow in the axial direction such that the exit planes of the pipes form an annular ring which exhausts the flow into the combustion chamber. A typical fish-tail diffuser installation at the exit of a centrifugal impeller is shown in Figure 2.2 .

Kenny (1968, 1972) tested a fish-tail diffuser and found its performance to be superior to the performance of vaneless and vaned diffusers. However, the operating range of the fish-tail diffuser was lower than that of the conventional designs. There exists a need for more detailed information regarding the flow within the fish-tail diffuser, to allow designers to understand the underlying physics governing these flows, and to improve their designs, if possible.

In the present work, a large scale model of a curved diffusing duct, approximately a portion of

Figure 2.1: The geometry and notation of the vaned diffuser, according to Dean (1971), reproduced from Cumpsty (1989).

Figure 2.2: The geometry and notation of the fish-tail diffuser installation.

a Pratt and Whitney Canada fish-tail diffuser, was designed and manufactured. The goal is to achieve detailed flow measurements throughout the diffuser for various spatial and temporal velocity distributions at the diffuser inlet. A detailed discussion of the test section is given in the following chapter.

2.1.4 Impeller Discharge Flow

The flow discharging from a centrifugal compressor is highly swirled, unsteady, turbulent, and transonic. Each of these factors can have significant effects on the performance characteristics of the diffuser.

The flow exiting the impeller was first described by the jet-wake model proposed by Dean and Senoo (1960), as shown in Figure 2.3 . The velocity in the wake is small relative to that of the jet, exhibiting lower relative stagnation pressure and higher turbulence levels. The model of Dean and Senoo assumes that flow non-uniformities are confined to the circumferential direction only. More recent investigations (eg. Eckardt, 1979 and Inoue and Cumpsty, 1984) have shown that significant flow non-uniformities exist in the hub-to-shroud direction, as seen in Figure 2.4 . According to Cumpsty (1989), the hub-to-shroud flow nonuniformity is a more important factor influencing diffuser performance.

The circumferential non-uniformity, in the impeller frame of reference, is responsible for unsteadiness in the diffuser passages. Inoue and Cumpsty (1984) found that these periodic temporal fluctuations were very strong at the passage entrance of vaned diffusers but decayed very rapidly and were virtually undetectable a short distance into the diffuser passage. In a separate study, the effect of rotor blade wakes on radial vaned diffusers was investigated by Baghdadi (1977). He found that blade wake mixing and flow unsteadiness had no significant effect on the performance of a vaned diffuser. He suggested that this was due to the rapid mixing of the wakes, which was accelerated due to differences in jet and wake flow angles, and lack of response of the diffuser vanes to the high frequency fluctuations caused by the rotating wakes.

Figure 2.3: The idealized jet-wake model. (Reproduced from Dean and Senoo, 1960)

Figure 2.4: The evolution of the flow pattern measured in a vaneless diffuser, according to Eckardt (1979), reproduced from Cumpsty (1989).

Experiments involving vaneless diffusers, such as those investigated by Eckardt (1975) and Inoue and Cumpsty (1984), have revealed that the radial extension of circumferential flow distortions is significant to a much greater extent than for vaned diffusers. Apparently, the presence of the guide vanes effectively mixes out the unsteady fluctuations in a very short distance, resulting in a steady flow through the majority of the diffuser passage.

Kenny (1968) has argued that one of the reasons for the superior performance of fish-tail diffusers is that they are able to redistribute the hub to shroud flow nonuniformity more effectively than vaned diffusers.

The present study focuses on a fish-tail diffuser design which is based on a P&WC fish-tail diffuser. The large scale of the diffuser will allow the detailed study of temporal and spatial variations of the flow in the diffuser. The diffuser test section inlet corresponds to the throat of the fish-tail diffuser on the engines. There is significant diffusion in the semi-vaneless space (shown in Figure 2.2) and the flow in this region is very complex. However, in the present study, only the diffusion downstream of the throat is studied to reduce the complexity of the test section.

2.2 Straight Diffusers

2.2.1 General Description

As mentioned earlier, very little work has been published on the detailed aerodynamics of curved diffusing ducts. However, a large body of work is available on the aerodynamics of diffusers with straight centerlines, such as conical, annular, and straight two-dimensional diffusers.

The geometry of these diffusers is defined through their exit-to-inlet area ratio, and the diffuser length non-dimensionalized by the inlet radius or inlet width. An alternative parameter, which is dependent on the area ratio and the length, is the diffuser wall angle.

2.2.2 Performance Parameters for Diffusers

Several different parameters are commonly used in the literature to measure the performance of diffusers. Since the primary function of diffusers is to convert dynamic pressure into static pressure, a natural parameter for expressing diffuser performance is the pressure recovery coefficient (C_p). The pressure recovery coefficient non-dimensionalizes the static pressure recovery by the mean inlet dynamic pressure as follows:

$$C_p = \frac{P_2 - P_1}{\frac{1}{2} \rho_1 \bar{U}_1^2} \quad (2.1)$$

where subscripts 1 and 2 refer to the inlet and outlet of the diffuser respectively. It can be shown that for one-dimensional, isentropic flow the pressure recovery coefficient is given by:

$$C_{pi} = 1 - \left(\frac{A_1}{A_2} \right)^2 \quad (2.2)$$

where A_1 and A_2 are the inlet and exit areas respectively.

The ideal pressure recovery coefficient can be used to form another common measure of diffuser performance, the diffuser effectiveness (or efficiency). The diffuser effectiveness is defined by:

$$\eta = \frac{C_p}{C_{pi}} \quad (2.3)$$

The total pressure loss is also used to measure diffuser performance. However, to calculate the total pressure loss, the velocity profiles at inlet and exit of the diffuser are required (Klein, 1989). A more convenient parameter, requiring velocity profiles at the inlet only, is known as the effective total pressure loss which is defined as follows:

$$\bar{H}_L = C_{pi} - C_p \quad (2.4)$$

2.2.3 Flow Regimes in Diffusers

Diffuser geometry is the main factor influencing the flow regime under which the diffuser operates, although other factors such as inlet velocity profile, and inlet turbulence intensity have noticeable influence as well. At small included angles and area ratios, the flow is steady and essentially free of any stalled regions. As the divergence and/or area ratio is increased, small unsteady regions of stalled flow develop and high frequency, small amplitude fluctuations in pressure occur. As the divergence and/or area ratio is increased further, large areas of stalled flow form, with corresponding low frequency, large amplitude pressure fluctuations. Maximum pressure rise normally occurs when some stall exists in a diffuser; however, significant regions of stalled flow severely compromise the pressure recovery and the outlet flow is very distorted with regions of reverse flow.

Among others, Reneau *et al.* (1967) provided extensive results on straight, two-dimensional diffusers. They found that four primary flow regimes could occur, depending on the area ratio and the included angle of the diffuser. The four flow regimes were: no appreciable stall regime, large transitory stall regime, two-dimensional stall regime, and the jet flow regime.

2.2.4 The Effects of Inlet Conditions on Diffuser Performance

Aside from diffuser geometry, the most important factor determining diffuser performance is the mean velocity profile at the inlet of the diffuser. Performance is decreased as the extent of low momentum fluid near the walls of the diffuser increases. Flow separation (or stall) takes place when the transfer of momentum to the near wall flow is no longer sufficient to move the flow against the adverse pressure gradient. Consequently, high inlet boundary layer shape factors and high inlet area blockage will in general lead to a loss in pressure recovery (Klein, 1981). Furthermore, any non-uniformities in the inlet velocity profile will be amplified in the diffuser resulting in a greater susceptibility to stall, compared to the case of uniform inlet flow. For area blockages caused by a velocity defect in the core flow, increases in pressure recovery are possible in some cases.

The inlet free-stream turbulence is also an important factor affecting diffuser performance.

Stevens and Williams (1980) studied the effect of inlet turbulence on the performance of annular diffusers. They found that an increase in the turbulence intensity improved the outlet flow stability and increased the pressure recovery by up to 20%, with only small increases in total pressure loss. Turbulent mixing tends to transfer momentum to the near-wall flow, thus reducing the shape factor of the boundary layers on the diffuser walls. Therefore, exit flow blockage is reduced, resulting in an increase in pressure recovery. Similar conclusions were reported by Hoffmann (1981) for two-dimensional diffusers, and by Bradley and Cockrell (1971) for conical diffusers.

Transition from laminar to turbulent flow in a diffuser has complicated effects on diffuser performance which are not well understood. However, for fully turbulent inlet flow, Reynolds number variation appears to have little effect on diffuser performance (Klein, 1981).

It is clear that inlet flow conditions to the diffuser are an important consideration when diffuser performance measurements are of interest.

2.3 Constant Area Curved Ducts

Curved ducts of constant area are extensively used in a wide range of applications, such as pipe networks, aircraft intakes, cooling coils of heat exchangers, and exhaust headers to name a few. The primary characteristic that distinguishes curved duct flows from straight duct flows is the generation of streamwise vorticity (or secondary flow) which results in pressure loss, spatial redistribution of streamwise velocity, and increased heat transfer at the duct walls. The primary geometric parameters affecting the flow development in the duct are the deflection angle, the ratio of the bend centerline radius to the bend cross-section hydraulic radius, and the cross sectional shape of the duct. Amongst others, Miller (1978) presents a thorough description of the effects of duct geometry on flow development in curved ducts. Taylor *et al.* (1982) investigated the effects of Reynolds number, velocity distribution, and turbulence on the flow development of curved ducts with rectangular cross section.

The secondary motion in curved ducts arises from the imbalance of the centrifugal force and radial pressure gradient at the side walls of the bend (Humphrey *et al.*, 1981). Centrifugal and pressure forces deflect the faster moving core flow towards the outside of the bend along the symmetry plane of the duct. Energy deficient fluid approaching the outer wall of the bend cannot pass through the adverse pressure gradient and is deflected along the side walls towards the low static pressure region at the inner wall of the bend. The movement of the core flow towards the outside wall along the symmetry plane, combined with the deflection of the low momentum fluid towards the inner wall along the side walls of the bend, results in a pair of counter-rotating streamwise vortices, as shown in Figure 2.5. For sufficiently high cross-stream pressure gradients, separated (or stalled) flow can occur on the inner wall at the exit of the bend.

2.4 Experimental Investigations of Diffusing Flow in Curved Ducts

As mentioned previously, very little work has been done on the combined effects of diffusion and flow path curvature on duct flows. The geometries are often complex and experimental design and testing is difficult and costly. Several experimenters, such as Wellborn *et al.* (1992), and Tindell (1987), have studied the flow in diffusing S-ducts. These S-ducts are common on several aircraft engine intakes such as the Boeing 727, General Dynamics F-16, and McDonnell-Douglas F-18. The present study is concerned with the flow through curved diffusing ducts without an inflection point.

Kenny (1968 and 1972), of Pratt and Whitney Canada, investigated the flow through fish-tail diffusers. As described in Section 2.1.3, these diffusers are used to diffuse and redirect the flow from the exit of the centrifugal impeller. The experiments were performed on 1:1 scale models which were not large enough to be instrumented for detailed internal measurements. Inlet and exit measurements were recorded and performance characteristics of the fish-tail diffuser were mapped for different operating conditions.

More recently, Blair and Russo (1980) of General Electric investigated a similar compact diffuser

Figure 2.5: Secondary flows in a curved duct. (Reproduced from Miller, 1978)

design for centrifugal compressors. As for Kenny, the apparatus of Blair and Russo was 1:1 scale and no internal measurements were taken.

Rojas *et al.* (1987) investigated the forced convective heat transfer in curved, rectangular diffusers. They obtained velocity distributions by laser-Doppler anemometry in a water tunnel and the distribution of the rates of heat transfer by liquid crystal thermography in a wind tunnel.

It is clear that there is a need for detailed measurements of the flow in curved diffusing ducts, particularly in those with small radius of curvature, for designers to understand the underlying physics governing the flow and to allow for improved designs in the future. The present study is concerned with the experimental investigation of the detailed flow development in a large scale fish-tail diffuser test section. Chapter three presents the details of the design and manufacturing procedure of the test section.

Design of the Fish-Tail Diffuser Test Section

3

As discussed earlier, there appears to be a lack of information on curved diffuser flows in the literature, particularly for curved diffusers with complex cross sectional geometries and small centerline radius of curvature such as those used in gas turbine engines. The flows through these diffusers are not well understood and the geometric complexity of the diffusers makes the design of test sections difficult and costly. The purpose of the present work is to design a large scale fish-tail diffuser test section.

This chapter describes the design of the fish-tail diffuser test section. The geometry of the flow path is given in Section 3.1. Section 3.2 gives a description of the inlet flow conditioning devices. Section 3.3 describes the geometry and design of the contraction upstream of the test section. Finally, Section 3.4 describes the overall design of the fish-tail diffuser test section, including the hardware and instrumentation.

3.1 Geometry of the Fish-Tail Diffuser

The original intention of the design was to accurately model the geometry of a Pratt and Whitney Canada (P&WC) fish-tail diffuser. Due to the complexity of the geometry, it was realized that an exact modelling would prove to be very costly due to manufacturing problems and instrumentation difficulties. In this section, a description of the P&WC fish-tail diffuser geometry is given. This is followed by a description of the simplifying changes to this geometry leading to the geometry of the diffuser flow path investigated in this study.

The P&WC fish-tail diffuser has several distinctive features. At the inlet of the diffuser the cross-section is circular and the centerline of the diffuser remains straight along a conical flow path. Downstream of the cone, the diffuser turns about 65° and the cross section becomes progressively more oblong, characterized by semi-circular ends attached by straight lines. The long axis of the cross sections progressively increases while the short axis decreases. Downstream of the 65° bend, the flow path is characterized by a straight centerline and the cross sections continue becoming increasingly oblong. Furthermore, the cross sections become twisted relative to each other by about 10° . The final portion of the flow path is the most complicated: it continues twisting for an additional 10° and continues turning to complete 90° of total turning in the diffuser. There is also about 40° turning in a plane orthogonal to the 90° turn. The exit plane of the diffuser is highly oblong having an aspect ratio of about 5.5. The cross sectional area of the diffuser varies approximately linearly with respect to the distance measured along the centerline. The schematic of a typical fish-tail diffuser flow path, taken from Raw *et al.* (1989), is shown in Figure 3.1.

The turning of 90° is required to turn the flow from the radial plane at the impeller exit to the axial direction prior to discharge into the combustion chamber. The additional turning of about 40° in the tangential plane is required to deswirl the flow - ie. to orient the exit plane of the diffuser such that its normal is aligned with the axial direction. The 20° of twist is necessary to align the long axis of the exit plane of the diffuser with the tangential direction. In this way, the combined exits of the diffusers form an annulus which discharges into the annular combustion chamber as seen in Figure 3.2. The geometry of the P&WC diffuser flow path has been

Figure 3.1: Flow path of a typical fish tail diffuser. (Reproduced from Raw *et al.*, 1989)

influenced not only by aerodynamic considerations but also by manufacturing and space constraint considerations.

Several aspects of the P&WC fish-tail diffuser geometry would make manufacturing of an exact scaled-up model difficult and costly. The test section was manufactured by computer numerically controlled (CNC) machines. The diffuser test section was split into top and bottom halves along the short axis and the flow path was machined out of each half. The details of the design and manufacture of the test section will be presented in Section 3.4 . The first difficulty with the P&WC geometry was the twisting of the cross sections which would result in 'negative' areas in the machining process. The machining tool would not have a free path to the duct contour as illustrated in Figure 3.3 . Furthermore, flow measurements would become unnecessarily difficult with the given instrumentation (to be discussed in detail in Section 3.4.5). The second difficulty with the P&WC flow path was the geometry of the exit portion of the fish-tail diffuser. As mentioned, this section has turning in two orthogonal directions and is further complicated by the high aspect ratio of the cross sections which make machining difficult.

Due to the above-mentioned difficulties, the P&WC fish-tail diffuser geometry was considerably simplified as shown in Figure 3.4 . Absence of the relative twist between cross sections, a simpler centerline path, and a straight "tailpipe" replacing the deswirl section of the original P&WC diffuser were some of the simplifications implemented. The diffusion rate and the radius of curvature of the radial-to-axial turn were chosen to be close to that of the original P&WC diffuser. The resultant flowpath of the fish-tail diffuser is shown in Figure 3.5 .

The above-mentioned modifications to the P&WC fish-tail diffuser geometry will certainly have aerodynamic consequences. This is not a major concern since the main focus of this project is not to specifically study the P&WC fish-tail diffuser but to understand the underlying physics involved in the flow through general curved diffusers with small turning radius. In fact, the present omission of the 40° deswirling section is in all likelihood more indicative of curved diffusers in other applications.

Figure 3.2: The installation of fish-tail diffusers on the centrifugal impeller

Untwisted Section
No Negative Areas

Twisted Section
Negative Areas

Figure 3.3: Illustration of negative areas which arise from twisted cross sections.

Figure 3.4: The area and cross sectional shape distribution for the fish-tail diffuser.

Figure 3.5: The geometry of the fish-tail diffuser flow path, including the tailpipe.

3.2 Establishing Inlet Flow to the Diffuser

As mentioned in Section 2.2.4, the inlet mean flow and turbulence characteristics are the dominant flow parameters influencing the performance of diffusers. Therefore, it is necessary to incorporate flow conditioning devices upstream of the diffuser inlet to control these inlet flow parameters.

A promising method for establishing a suitable mean velocity profile and turbulence structure is by using vortex generators. Bradley and Cockrell (1971) employed this method for establishing inlet flow conditions to a conical diffuser and found satisfactory results. The system consisted of an annular step used to create a momentum deficit followed by eight triangular vortex generators set at alternating incidences of $\pm 5^\circ$. Immediately downstream of the vortex generators the pipe wall was roughened for a length of six diameters.

In the current design, the dimensions for the vortex generator system are directly taken from the work of Bradley and Cockrell (1971). The pipe used to house the vortex generators is made of poly-vinyl chloride (pvc). The pipe is durable, smooth and has relatively close tolerances although the internal finish was found to be somewhat wavy. The inside diameter of the pipe was machined to 7.625 inches to eliminate the wavy surface finish. The pipe length is 60 inches and it was necessary to segment it into three equal parts since the largest available lathe was not capable of machining the entire length. The three pieces are then clamped together. Additional flanges were machined in order for the pipe to be attached to the diffuser test section in 1/3 and 2/3 length configurations.

The pipe is instrumented with two traversing locations, 2.75 inches downstream of the pipe inlet and 3.00 inches upstream of the pipe exit. The first location is to be used for assessing the level of flow uniformity developed by the contraction and the second location is used to determine the inlet flow properties to the diffuser established by the inlet flow conditioning devices. Pitot probes are used to measure the velocity profile. A seven-hole probe is used to determine flow angularity. Finally, to measure turbulence quantities, a hot wire anemometer is used.

Both ends of the pvc pipe are fastened to aluminum flanges by 6 fasteners. The flanges can be fastened through 12 equally spaced holes. The purpose of the small increment is to facilitate

Figure 3.6: Vortex generator system used to pre-condition the flow upstream of the diffuser inlet.

rotation of the end segments of the pipe independently at 30° intervals in order to traverse different radial paths with the same traversing gear. Since the pipe is segmented, it is possible to rotate each section of the pipe independently. A perspective view of the system is shown in Figure 3.6 .

As mentioned in Section 2.1.4, the flow entering the diffusers in gas turbine engines is unsteady, nonuniform, and highly turbulent. Currently, the fish-tail diffuser test rig is being used to study the effects of various symmetric and asymmetric mean flow distributions at the inlet which correspond to different types of throat blockages in the engine fish-tail diffuser. The conditions were generated by adjusting the pipe length and installing rings at the pipe inlet.

Additional tests are underway with flow conditioning devices that will significantly alter the turbulence intensity at the diffuser inlet. The vortex generator system, which was described above, and an alternate system such as a perforated plate will be used. Future tests will incorporate the effects of flow unsteadiness on the performance of the diffuser.

3.3 Wind Tunnel Contraction Design

Contractions are commonly used upstream of test sections in wind tunnels to establish steady, parallel, uniform flow with low turbulence. For the present application, these characteristics are desirable; however, the contraction is a requirement also on geometric grounds. The pvc pipe that feeds the diffuser has an inside diameter of 7.625 inches. The diameter of the wind tunnel flange downstream of the settling chamber is 16 inches. Therefore a contraction area ratio of 4.4 is required. The inlet and exit diameters are fixed and it is necessary to determine the appropriate contraction length and contour.

The main challenge in contraction design is to avoid flow separation which is caused by adverse pressure gradients experienced near the entrance and exit of the contraction. Additionally, possible turbulent boundary layer relaminarization near the exit increases the likelihood of flow

separation in this region. Flow separation will lead to a degradation of the exit flow and may cause unsteadiness in the case of large scale separation.

Until the early 1970's, contraction design was usually based on the solution of the incompressible, inviscid flow equations such as Laplace's equation with little regard to the physical processes involved. Performance of contractions is usually governed by viscous effects and limited by flow separation. Therefore, by using inviscid analyses it is impossible to determine optimum nozzle shape and length. Since then, several researchers have performed comprehensive studies on wind tunnel contraction design. Chmielewski (1974) specified a streamwise acceleration distribution to define a four parameter family of contraction cones which offered the minimum length without experiencing boundary layer separation. Stratford's criterion (Stratford, 1959) for turbulent separation was used to control the severity of adverse pressure gradients on the contraction wall. Morel (1975) determined contraction contours from two cubic arcs joined smoothly together at the inflection point. Design charts are given to establish the contraction length and position of the inflection point based on the required degree of exit flow uniformity and allowable pressure coefficients at inlet and exit. Mikhail and Rainbird (1978) developed a numerical scheme based on the method of lines supplemented by Stratford's criterion for turbulent separation. Their method related the shape and length of the contraction to the flow quality required in the test section and to the contraction ratio. They found that this method could reduce contraction lengths of previous designs by as much as 50% while maintaining the same level of flow quality in the test section. Being very easy to use, the method established by Morel was adopted in the present study. Furthermore, confidence in this method has been gained due to the successful contraction design of Klein (1989) at Carleton University.

The family of contours described by Morel are defined by one parameter,

$$X = x_m / L \quad (3.1)$$

where L is the length of the contraction and x_m is the location of the match point between the two cubic curves as shown in Figure 3.7. As already mentioned, the design criteria for wind tunnel contractions are exit flow uniformity, avoidance of separation, and exit flow boundary layer thickness. The controlling geometric variables are the wall shape and duct length. By using the design charts of Morel (1975) it was determined that the optimum length for the contraction was

Figure 3.7: Wall contour constructed of two matched cubic arcs.

Figure 3.8: The geometry of the contraction flow path.

$L=15.535$ inches and the location of the match point was defined by $X=0.346$.

The equations describing the contraction contours in terms of radius are:

$$r(x) = \left[1 - \frac{1}{X^2} \frac{x^3}{L^3} \right] (r_1 - r_2) + r_2, \quad \text{for } x < x_m \quad (3.2a)$$

$$r(x) = \frac{r_1 - r_2}{(1 - X)^2} \left[1 - \frac{x^3}{L^3} \right] + r_2, \quad \text{for } x > x_m \quad (3.2b)$$

Note that these equations are different than those found in Morel (1975), presumably due to a typographical error in his paper. The resulting contour shape is shown in Figure 3.8 and a table of the coordinates described by Equations 3.2a and 3.2b is given in Appendix 1A.

To manufacture the contraction, it was first necessary to manufacture a mandrel out of laminated wood. The mandrel was then used as the mold over which heated plastic was wrapped by a vacuum forming process. The mandrel was then removed and the plastic nozzle was fitted with aluminum circular flanges on either end.

3.4 The Fish-Tail Diffuser Test Section

3.4.1 Material

As will be discussed in Section 3.4.2 in more detail, the fish-tail diffuser flow path, with the exception of the conical section, was manufactured by a computer numerically controlled (CNC) machine. The material from which the flow path was machined is known as MB1000. MB1000 was specifically developed for precision machining and modelling. MB1000 is similar in appearance, texture and density to wood. Unlike wood, MB1000 is an isotropic block material (ie. no grain) and has superior dimensional stability and much lower water absorption than wood. The physical properties and test data for MB1000 are listed in Appendix 1B along with the corresponding values for Renshape, another modelling material which was considered.

During machining, it was found that the MB1000 material was prone to chipping if high displacement speeds were used for the tool or if more than about 0.03 inches of material was removed in each pass. The chipping would occur just before the tool cleared the material. The material was also found to be very abrasive. This required low tool speeds when machining the external geometry. Despite the low tool speeds, frequent sharpening of the tool was necessary. Similar problems were encountered when machining the flow path. Eight roughening tools and eight finishing tools were required to machine the flow path. Although MB1000 has better physical properties than Renshape and is comparatively priced, Renshape is claimed to be less abrasive and easier to machine.

MB 1000 comes in boards of 2"x16"x60" which are pre-planed on the top and bottom. For the present application, it was necessary to laminate the boards together with an epoxy resin, BC-5003. BC-5003 cures in 6 to 8 hours at room temperature (75° F). A description of the physical properties of BC-5003 is given in Appendix 1B. To reduce the quantity of material needed, the MB1000 boards were cut to size and laminated into channel sections. In this way, the entire test section required only 11 boards of the material. A further advantage of laminating the boards into channel sections was to reduce the machining time and cost since less material had to be removed when the flow path was milled out of the sections. A typical half section is shown in Figure 3.9.

3.4.2 Machining of the Diffuser Test Section

Once the lamination of the channel sections was complete, the sections were externally shaped on a milling machine by the Engineering Technical Centre - Mechanical (ETC) at Carleton University. Due to the size of the large sections, it was necessary to machine them in two passes by shifting the sections sideways. Therefore, it was necessary to relocate the reference points and to align the sections properly whenever they were shifted. Also, a custom made sine-table was used to machine the top and bottom surfaces of the sections to the specified angles. Once the external surfaces were machined, flanges and traversing gear mounting plates (which are further described in Section 3.4.6) were installed.

The flow path of the conical section was machined on a lathe at ETC. The two half sections were clamped together and mounted to the lathe with the inlet side (ie. smaller diameter side) closest to the chuck to allow for tool access. This procedure posed several difficulties. Clamping the two

Figure 3.9: Lamination of a typical half section prior to machining.

halves together under pressure slightly deformed the section under this load. When the load was removed, the flow path was not an exact circle. However, the resultant path was reasonably acceptable although the transition from the conical section to the downstream section was not as smooth as the transition between the other sections. Another reason for this relatively rough transition was caused by the difficulty of aligning the center of the exit opening with the center of the lathe.

The flow paths of the remaining sections were milled by a computer numerically controlled (CNC) machine at the Science Technology Centre (STC) at Carleton University. The geometry of the diffuser flow path was produced three dimensionally using CADKEY version 4.1, a general purpose computer aided design (CAD) program. Figure 3.10 illustrates the flow path mesh produced by CADKEY. The geometry is defined by straight lines, circular arcs, and cubic splines. The CAD file, produced by CADKEY, was given to STC where it was translated to a computer aided manufacturing (CAM) file. This CAM file was used as an input deck for programming the tool paths. The programming required 35 man hours while the actual machining of the flow path required 110 man hours. An additional 100 man hours were required for the setup and alignment of the sections and tools. Once the CNC machining was complete, 23 man hours of hand finishing was necessary to smooth the surface of the flow path.

3.4.3 Partitioning of the Diffuser Test Section

The largest CNC machine at STC supports parts no bigger than $17\frac{1}{2}'' \times 17\frac{1}{2}'' \times 27\frac{1}{2}''$. Due to the scale of the diffuser test section, it was necessary to divide it into four longitudinal sections. Each section was further divided into symmetric top and bottom halves in order to mill out the flow path.

This partitioning has also provided easy probe access to any cross section throughout the diffuser. The longitudinal partitioning was done as follows:

- Section 1: Conical section 19" in length. Restriction of the conical portion of the flow path to one section allowed machining by a lathe.
- Section 2: Oblong cross sections begin; 32° of turning is completed after a short

Figure 3.10: Diffuser mesh, produced by CADKEY, used for CNC machining.

Figure 3.11: Illustration of the fish-tail diffuser test section.

straight segment. The length of the centerline is about 23".

Section 3: Continued turning for an additional 32° over a centerline arc length of about 18".

Section 4: Additional 26° of turning to complete the 90° turn. Centerline length is about 16".

The assembled test section is illustrated in Figure 3.11 .

3.4.4 Tailpipe

As mentioned in Section 3.1, a straight tailpipe, 25" in length, was fitted to the outlet of the diffuser. The tailpipe cross section is constant and is identical to the cross section of the outlet of the diffuser flow path. Without the tailpipe, there would likely be some flow deviation at the outlet since the diffuser flow path completes the 90° turn at the exit plane of the diffuser. The tailpipe was manufactured from galvanized sheet metal, 1.32 inches thick, bonded to an aluminum frame. The installation of the tailpipe to the diffuser test section is shown in Figure 3.11.

3.4.5 Instrumentation

3.4.5.1 Seven-Hole Probe Measurements

The flow field in the diffuser is expected to be three-dimensional in which flow angles will not be known *a priori*. Flow three-dimensionality at the diffuser exit is a concern in many curved diffuser applications; for example, in the case of fish-tail diffusers it affects the combustion process. The flow three-dimensionality throughout the diffuser also affects the loss generation process and the velocity distortion at the exit and hence the static pressure rise in the diffuser.

For applications where the mean flow direction at a given point is unknown, a multi-hole probe is recommended. A seven-hole probe will be used on the fish-tail diffuser rig. Due to the small dimensions of the seven-hole probe (0.083 inches diameter) relative to the scale of the flow path, the flow should not be significantly affected by the presence of the probe.

The seven-hole probe consists of seven pressure sensing stainless steel hypodermic tubes (inside diameter: 0.0105"). One tube is located in the centre with the other six arranged around it. The bundle of seven tubes is housed in a stainless steel tube casing. At the tip of the probe the space

Figure 3.12: Construction of the seven-hole probe.

Figure 3.13: Near-wall measurements by the seven-hole probe

between the seven tubes is filled with solder and the tip is tapered to a 30° semi-apex angle as shown in Figure 3.12 .

The seven-hole probe was designed with an s-bend in order to measure the flow close to the diffuser walls as shown in Figure 3.13 . The probe is mounted in a probe holder which is attached to an adjustable probe stem. The probe stem consists of two separate bars attached by a 90° elbow. This configuration allows adjustment in both vertical and horizontal directions. An auxiliary spring loaded stem supporter can be used when the probe is traversing the bottom of the flow path. This extra support is necessary to avoid vibration of the stem due to aerodynamic loading. The seven-hole probe and hardware is shown in Figure 3.14.

Alignment of the probe axis relative to the cross sections which are being traversed is vital for taking reliable measurements. An alignment apparatus (shown in Figure 3.15) is used to properly align the probe in both horizontal and vertical directions. The alignment is performed by removing a diffuser upper half-section and placing the bottom of the alignment apparatus on the top/bottom interface flange of the lower half-section. The front of the alignment apparatus is then butted up against the next upstream section. A dummy probe mounted on the traversing apparatus on the upstream section is then aligned with the correctly positioned pointer. When the alignment is complete, the dummy probe is replaced by the seven-hole or hotwire probe.

The pressure sensed by each hole of the probe is measured by a transducer relative to a reference pressure. By relating these pressures to known calibration curves of the probe, the static pressure, total pressure and flow angles can be determined. During testing this will all be done automatically with a data acquisition system.

3.4.5.2 Hotwire Measurements

As mentioned in Section 3.2, extensive turbulence measurements in the test section are being planned. Also, future studies of unsteady flow through the diffuser are anticipated. The only plausible instrument for such measurements in this test section is the hot wire anemometer. Laser anemometry is not suitable since transparent windows would be necessary in the test section for the laser and optics to 'see' the flow.

Figure 3.14: The seven-hole probe and traversing gear unit.

Figure 3.15: Alignment apparatus for the seven-hole and hotwire probes.

Hot wire anemometers measure fluid velocity by sensing the changes in heat transfer from a small electrically-heated wire exposed to fluid motion. For the current study, a four-wire anemometer will be used which will be capable of measuring the three components of velocity. The same traversing gear and probe holder as for the seven-hole probe will be used by the hot wire probe.

3.4.5.3 Traversing Gear

A motorized traversing gear unit is used to control the position of the probe automatically along a one dimensional path. Different settings of the horizontal probe stem are then used for traversing different vertical paths in the measurement plane.

A four phase stepping motor is used to drive the traversing gear, which in turn is driven by signals from an external controller. The rotation of the motor is transferred to linear motion by a 1/4"-20 threaded rod. The gear provides linear motion with an estimated accuracy of $\pm 0.1\text{mm}$. The traversing gear unit is shown along with the seven-hole probe in Figure 3.14 .

3.4.6 Hardware

3.4.6.1 Mounting Plates

The traversing gear unit is fastened to the rig on mounting plates. A typical mounting plate is shown in Figure 3.16. Four mounting plates are provided for each section with the exception of section 1 which only has two. Two mounting plates are located on top of sections 2 to 4 as shown in Figure 3.11. The other two mounting plates are located on the side wall of the bottom half of each of sections 2 to 4. A typical example is shown in Figure 3.17. Section 1 has one mounting plate on the top and one mounting plate on the side wall. The mounting plates located nearest to the inlet of each section are placed such that the probe traverses the interface planes between adjacent diffuser sections. The additional mounting plates allow probe access to traversing planes approximately in the middle of each section. Through the use of more elaborate probe stems, probe access to other measurement planes throughout the diffuser would be possible.

The top mounting plates were machined to an angle such that the probe stem is vertical (ie. aligned with the z-axis). The mounting plates located on the side walls of each section were

Figure 3.16: The geometry of a typical mounting plate.

Figure 3.17: Installation of mounting plates on a typical lower half-section.

machined to an angle such that the probe stem would be aligned horizontally (ie. aligned with the y-axis).

3.4.6.2 Flanges

The individual diffuser sections are flanged together with angle aluminum bars having dimensions of 1 1/4" x 1 1/4" x 1/4". The purpose of the flanges is not only to fasten the diffuser sections together but also to align the sections and to prevent leakage between mating surfaces. In order to accomplish alignment and efficient sealing, the flanges were machined with either 'male' or 'female' grooves so that adjoining flanges would be mated, as shown in Figure 3.18.

Alignment was of paramount importance since any misalignment would result in small steps in the flow path between diffuser sections. These steps could initiate boundary layer separation thus contaminating the flow field. Due to the tolerances of the angle aluminum flange mounting holes, it was decided that 1/4 inch stainless steel dowels would be used to accurately align the flanges to the diffuser sections themselves. Once accurate location of the flanges was assured, it was possible to accurately align the sections to each other.

Each section was fitted with either 'male' or 'female' flanges. For the sections with 'female' flanges (ie. Sections 2 and 4) it is possible to remove the top half of the section without disassembling the rig. For the other sections, disassembly is necessary before removal of the upper half-section.

3.4.6.3 Structural Support

The completed test section is relatively heavy and it was necessary to design a structural support frame to rigidly carry the load. Heavy angle iron bars were used to construct the bulk of the frame. The test section is supported at each diffuser section interface by the aluminum flanges which are placed and secured with set screws in iron channels.

The whole system can easily be moved since castors are integrated into the stand. The floor in the turbomachinery lab may have a slight slant to it. Therefore, the castors were attached to the

Figure 3.18: 'Male' and 'female' type flanges used for alignment and sealing between diffuser sections.

frame by 1 inch steel threaded studs. In this way, the height of each castor can be adjusted to level the test section such that the symmetry plane of the diffuser is horizontal. The complete system is shown in Figure 3.11 .

Future Goals

4

As mentioned in Section 3.2, several experiments on the fish-tail diffuser test section have been conducted thus far. The effects of various axi-symmetric and asymmetric inlet mean flow distributions on the performance of the fish-tail diffuser have been studied. Measurements have been taken with the seven-hole probe at the exit plane of the diffuser (ie. at the inlet plane of the tailpipe) to determine the overall performance of the diffuser in terms of pressure recovery, total pressure loss, and exit flow uniformity. No measurements have been taken between the inlet and exit of the diffuser thus far.

Experiments that will focus on the effects of various inlet turbulence intensities on diffuser performance are underway. Future experiments will study the effects of unsteadiness. A four-wire hot wire anemometer will be used to measure the temporal velocity field in these experiments. Further studies will focus on the flow structure within the diffuser. Measurements will be taken with the seven-hole probe and hot wire probe in several cross sections along the

diffuser providing insight into the development of the three-dimensional flow.

The current test section is used to study incompressible flow conditions. This is not representative of the Mach numbers found in gas turbine fish-tail diffusers. The recent commissioning of the high speed wind tunnel facility at Carleton University allows the future possibility to test a smaller scale model of the fish-tail diffuser at high Mach numbers.

Numerical simulation of the fish-tail diffuser flow is also planned. These computational studies will be done with a three-dimensional Navier Stokes solver being developed at Carleton University. The goal is to identify an efficient computational algorithm and a suitable turbulence model for this flow, thus providing future designers of such diffusers with a validated tool for affordable design optimization.

PART II:

**Computations of the Effects
of AVDR on
Transonic Turbine Cascades**

gaining popularity as grid generators for finite-element and finite-volume methods. The most recent version of Dawes' code uses unstructured grids (Dawes, 1992 and 1993). Solution-adaptation implies that the grids are refined automatically as the solution progresses in regions that require higher resolution to capture the local flow phenomena accurately. The main advantages of unstructured grids are as follows:

- (i) Greater geometrical flexibility allowing for geometries of arbitrary complexity and high resolution in local regions without the necessity for global refinement (unlike structured methodologies).
- (ii) Well suited for the implementation of grid adaption (as a function of both time and space) in the form of grid enrichment or depletion in areas experiencing strong gradients or high truncation errors; this allows for user independent solutions since no *a priori* knowledge of the solution is required for the construction of the optimal mesh.
- (iii) High degrees of accuracy are possible by refining the mesh since small local elements will increase the local accuracy without the risk of developing badly deformed or skewed elements.

Unstructured grids are still in the development stage but have recently been implemented for turbomachinery analysis, for example: Dawes (1992,1993) and Siden *et al.* (1990). Most of the codes using unstructured grids are currently used in research environments and are not yet competitive with conventional structured techniques. Typically, to achieve comparable accuracy and detail takes about three times as long with the unstructured mesh than with the structured mesh due to increased computer storage requirements for the unstructured mesh. In general, a three-dimensional unstructured solution adaptive mesh is very complicated and can take far more cpu time to generate than the actual flow simulation (Dawes, 1992). An example of an unstructured grid for a turbine blade analysis is given in Figure 6.2 .

complicated flows such as those involving turbulent, chemically reacting, or separated flows, particularly around complicated geometries, are beyond current capabilities and presently rely on empiricism. The current stage of advancement allows solutions where "problems involving complex geometries can be treated with simple physics and those involving simple geometry can be treated with complex physics" (Fletcher, 1988a). CFD testing complements rather than replaces experimental testing. CFD testing is useful for reducing the range of conditions over which experimental testing is required. Experimental testing will always be necessary to validate CFD codes especially when new machines or geometries are being investigated.

CFD analysis for air-breathing propulsion systems is typically more complicated than for their external aerodynamic counterparts in three principal ways, as outlined by Barber (1987):

- (i) Propulsion systems usually involve complicated geometries.
- (ii) There is a close coupling of aerodynamic components involving three-dimensional unsteady and viscous effects.
- (iii) There is a significant energy exchange from either chemical reactions or mechanical work.

Any attempt to accurately simulate the flow field in a typical propulsion system must rely on the numerical solution of the Navier-Stokes equations. A substantial amount of physical, numerical, and hardware development is necessary before Navier-Stokes solvers become efficient, accurate, and reliable engineering tools (Hanel, 1991). Even for the straightforward problem of calculating the loss of a two-dimensional cascade, Denton (1993) maintains that an *a priori* prediction using the best available methods is unlikely to be accurate to better than about $\pm 20\%$. For such cases, CFD analysis can be used as a preliminary design tool or to establish qualitative trends in performance, but resolution of flow details or accurate quantitative predictions are still some way in the future.

* * * * *

The highly competitive gas turbine business has led designers to strive for ever higher efficiencies, lower emissions, and lower noise from their engines. A few efficiency points can mean the difference between success and failure of any particular engine in this competitive market. Improvements in performance can be realized by careful design of aerodynamic components. To

make such improvements, it is necessary for the designer to understand the physical processes taking place in the turbomachinery component so that the sources of loss can be identified and minimized. Common sources of loss in axial turbomachinery blade rows include tip-leakage flows, boundary-layer separations, shocks, and trailing-edge effects.

Before understanding the details of three-dimensional flow phenomena in turbomachinery components, it is necessary to have a good understanding of two-dimensional flows. Two-dimensional testing is usually accomplished through the use of a linear cascade of blades. Linear cascades are typically used in research to examine blade optimization, shock/boundary-layer interaction, trailing-edge flows, and wake mixing. Since all real cascades are of finite span, they are inherently three-dimensional due to the development of boundary layers on the cascade endwalls. Although "two-dimensional" tests are generally performed on linear cascades of relatively low aspect ratio (1 to 3) and measurements are taken at midspan, three-dimensionality is present and is usually not accounted for. There is no general agreement regarding the tests necessary to establish two-dimensional flow nor for the degree of deviation from two-dimensionality which is acceptable.

One approximate measure of flow two-dimensionality is the axial velocity density ratio (AVDR). AVDR measures the acceleration or deceleration of the flow at midspan. A full description of the AVDR is found in Chapter 6. Flow acceleration in cascades with solid parallel endwalls is caused by the convergence of the blade channel due to blockage effects such as the displacement of the endwall boundary layer and secondary flow effects. For such cases, the AVDR can be as high as 1.1, whereas for ideal two-dimensional flow conditions the AVDR would measure 1.0.

The flow acceleration at midspan can affect the midspan performance of the blades. Several studies have been performed to determine the effects of flow convergence or divergence on the midspan performance parameters of compressor cascades, however very little work has been done on turbine cascades. Studies performed by Rodger (1992) were made on a low speed, variable incidence, linear cascade of turbine blades. Rodger found that flow convergence had less effect than flow divergence on total pressure losses. However, the trailing-edge deviation was significantly influenced by both conditions. Also, the base pressure at the trailing edge was found to be sensitive to the AVDR. Rodger thus speculated that cascades with large trailing-edge losses,

such as those operating at transonic conditions, would be more susceptible to lack of flow two-dimensionality.

The present study computationally investigates the effect of AVDR on the midspan performance parameters of turbine blades in cascade operating at transonic Mach numbers. The computational simulation was performed using a three-dimensional Navier-Stokes solver (BTOB3D) developed by Dawes (1988). The motivation of the study is two-fold: to evaluate the usefulness of the solver as an analysis tool for turbomachinery research and to extend the work of Rodger to transonic Mach numbers. Chapter 6 presents a brief overview of the terminology and technology used in today's Navier-Stokes solvers as well as a review of the work that has been done on AVDR studies. Chapter 7 gives a detailed description of the Dawes code including suggestions for improvement. A discussion of the mesh refinement study is also presented. The computational results are presented in Chapter 8 and conclusions and recommendations are presented in Chapter 9.

Theory and Literature Review

6

6.1 Computational Fluid Dynamics for Internal Flows

Computational fluid dynamic (CFD) analysis has increasingly grown in importance as a design or analysis tool for studying a wide range of internal flows. The physical processes governing these flows vary in complexity; they may involve compressibility, turbulence, system rotation, shocks or boundary-layer separations. Several factors must be considered when selecting or constructing a computational model for a particular problem:

- (i) The nature of the problem, including the physical processes involved and the equations necessary to adequately describe these processes.
- (ii) The degree of accuracy required: quantitative results or qualitative trends.
- (iii) The availability of resources in terms of computational speed and capacity.

Proper identification of the problem and consideration of the available resources will lead to appropriate choices regarding the nature of the computational model:

- (i) Selection of the equations governing the flow processes in question.
- (ii) Selection of the discretization scheme for the governing equations along with the method of solution.
- (iii) Selection of a turbulence model (if required) to close the system.
- (iv) Selection of a mesh strategy to enable accurate representation of the discretized equations.

A wide range of choices is available for each of the above points. The flow of interest in the present study is viscous, compressible, turbulent and three-dimensional. This limits the equations of motion to some form of the Navier-Stokes equations. The next section will discuss these equations and the main simplifications which can be applied to them. This is followed by a review of the methods available for discretizing the equations, the possible choices for the turbulence model, and the possible grid configurations. Emphasis will be placed on those options which were chosen in the development of the code used in the present work, namely the Dawes code (eg. Dawes, 1988).

6 1 Governing Equations for Fluid Flow

6.1.1.1 Full Navier-Stokes Equations

The physics of a compressible, continuum flow is completely described by the Full Navier-Stokes (FNS) equations. These equations are a mixed set of elliptic-parabolic equations for steady flow and a mixed set of hyperbolic-parabolic equations for an unsteady flow. The steady equations are difficult to solve due to the differences in numerical techniques used for hyperbolic and elliptic type equations (eg. Anderson *et al.* 1984). Therefore, the time-dependent equations are used for both steady-state and transient predictions. For steady-state solutions, the time-dependent equations are time-marched from an initial guessed solution until steady-state is achieved. The intermediate solutions have no physical meaning and the time step is limited by a stability criterion. For transient solutions, the solution is initiated from a known data set. The time step must be consistent with the physical requirements of the problem as well as satisfying the stability requirements.

The FNS equations are a statement of conservation of mass, momentum and energy. The rate of change of the conservation variables, U , in a control volume are balanced by the effect of generalized fluxes. The primary advantage of using the conservation form of the governing equations is that flow discontinuities such as shock waves are automatically treated since the conservation variables are continuous through shocks unlike the primitive variables (eg. u , v , w , p). In Cartesian coordinates and neglecting body forces and external heat addition, the FNS equations are written in vector form as follows:

$$\frac{\partial U}{\partial t} + \frac{\partial(E_i + E_v)}{\partial x} + \frac{\partial(F_i + F_v)}{\partial y} + \frac{\partial(G_i + G_v)}{\partial z} = 0 \quad (6.1)$$

where the conservation quantities are given by vector U and the flux terms in the three coordinate directions are separated into viscous and inviscid parts given as vectors E , F , and G :

$$U = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ E_t \end{bmatrix}$$

$$E_i = \begin{bmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ \rho uw \\ (E_t + p)u \end{bmatrix} \quad E_v = - \begin{bmatrix} 0 \\ \tau_{ux} \\ \tau_{vx} \\ \tau_{wx} \\ \tau_{hx} \end{bmatrix}$$

$$F_i = \begin{bmatrix} \rho v \\ \rho uv \\ \rho v^2 + p \\ \rho vw \\ (E_t + p)v \end{bmatrix} \quad F_v = - \begin{bmatrix} 0 \\ \tau_{vx} \\ \tau_{vy} \\ \tau_{vz} \\ \tau_{hv} \end{bmatrix}$$

$$G_i = \begin{bmatrix} \rho w \\ \rho uw \\ \rho vw \\ \rho w^2 + p \\ (E_t + p)w \end{bmatrix} \quad G_v = - \begin{bmatrix} 0 \\ \tau_{wx} \\ \tau_{wy} \\ \tau_{wz} \\ \tau_{hw} \end{bmatrix}$$

The viscous stress and heat conduction terms are given by

$$\tau_{xx} = 2\mu u_x + \lambda(u_x + v_y + w_z)$$

$$\tau_{yy} = 2\mu v_y + \lambda(u_x + v_y + w_z)$$

$$\tau_{zz} = 2\mu w_z + \lambda(u_x + v_y + w_z)$$

$$\tau_{xy} = \mu(u_y + v_x) = \tau_{yx}$$

$$\tau_{xz} = \mu(u_z + w_x) = \tau_{zx}$$

$$\tau_{yz} = \mu(v_z + w_y) = \tau_{zy}$$

$$\tau_{hx} = u\tau_{xx} + v\tau_{xy} + w\tau_{xz} + kT_x$$

$$\tau_{hy} = u\tau_{yx} + v\tau_{yy} + w\tau_{yz} + kT_y$$

$$\tau_{hz} = u\tau_{zx} + v\tau_{zy} + w\tau_{zz} + kT_z$$

In addition the thermodynamic variables are related by.

$$E_t = \rho \left[e + \frac{1}{2}(u^2 + v^2 + w^2) \right] \quad H = h + \frac{1}{2}(u^2 + v^2 + w^2)$$

$$p = \rho RT \quad H = \frac{1}{\rho}(E_t + p)$$

where E_t , e , H , and h are the total energy, specific internal energy, stagnation enthalpy, and static enthalpy respectively. For a perfect gas

$$h = C_p T \quad e = C_v T \quad C_p = \frac{\gamma R}{\gamma - 1} \quad C_v = \frac{R}{\gamma - 1}$$

An assumption of adiabatic flow simplifies the set of equations and replaces the energy equation by $H = \text{constant}$. In most cases the bulk coefficient of viscosity is negligible (ie. Stokes'

hypothesis) and the second coefficient of viscosity is given as $\lambda = \frac{2}{3}\mu$ (eg. Anderson *et al.*, 1984).

To account for turbulence, the equations are time averaged giving rise to the *Reynolds averaged* Navier-Stokes equations. These equations have the same form as the laminar equations and thus utilize the same solution techniques. By using the Boussinesq approximation, turbulence is taken into account by replacing the coefficient of viscosity μ by $\mu + \mu_t$ and by replacing the coefficient of thermal conductivity k by $k + k_t$. The terms with subscript t are the turbulent analogues to the laminar coefficients. These terms are determined from additional relationships, known as turbulence models, which then close the system. Turbulence modelling is discussed in more detail in Section 6.1.5.

The FNS equations are often written in nondimensional form. In this way the flow variables are normalized to fall between certain prescribed limits such as 0 and 1. The characteristic parameters such as Mach number, Reynolds number and Prandtl number then appear in the equations and can be varied independently (eg. Anderson *et al.*, 1984).

The FNS equations completely describe the physics of the flow field. Unfortunately, these equations are very difficult to solve in their complete form, requiring a large amount of computer time and storage for solution. Fortunately, for many viscous flow problems certain terms can be omitted thus reducing the FNS equations to simpler form. Two examples of reduced forms of the FNS equations are presented next.

6.1.1.2 Thin-Layer Navier-Stokes Equations

For viscous flows at high Reynolds numbers where there is an identifiable main streamwise direction, it is often satisfactory to use the Thin-Layer Approximation to the Navier-Stokes equations. Similar to the boundary-layer equations, all viscous terms with streamwise derivatives are neglected, but unlike the boundary-layer equations, all inviscid terms and time derivatives are retained. Consequently, streamwise diffusion is neglected but transition from the viscous dominated regions to the inviscid regions outside the boundary layer is an integral part of the calculation. The simplifications are valid for calculating mild separations and reverse flow regions

but begin to break down when the viscous regions become too large as would be the case in regions of large separation, in zones of shock-wave/boundary-layer interaction or in the base pressure region (Hirsch, 1985).

A further motivation for using the Thin-Layer Navier-Stokes (TLNS) equations arises from the typical grid configurations used in many high Reynolds number computations (Baldwin and Lomax, 1978). Grid step sizes in the streamwise direction are much larger than in the wall normal direction due to the disparity in length scales that must be resolved: the body length L and the boundary-layer thickness δ . In these computations, a substantial amount of computer expense is needed since highly stretched grids are necessary for resolving the normal gradients in the boundary layers. Consequently, the streamwise gradients are usually not adequately resolved even though the corresponding viscous terms are retained in the computation. Hence, for many computations it is reasonable to omit the viscous terms with streamwise derivatives since they are not being adequately resolved in any case.

The TLNS equations have been extensively used for investigating turbomachinery cases. Nakahashi (1989) used the reduced equations to study the flow field through a turbine cascade. Chima (1991) studied the flow through a rotating transonic fan rotor. Both studies concluded that the TLNS equations gave good predictions of experimental results. For the more complicated problem of flow through a centrifugal compressor, Chima (1987) solved the TLNS equations by employing a quasi-three-dimensional analysis in a body-fitted coordinate system and found favourable results. Although the results seemed to be acceptable in these cases, it is doubtful whether the reduced equations would be suitable for studying turbomachinery flows at large off-design incidences where large separation zones would begin to dominate the losses. The code used in the present study (Dawes, 1988) uses the Thin-Layer approximation and has been used with favourable results for a wide variety of turbomachinery cases as will be mentioned in the following chapter.

6.1.1.3 Parabolized Navier-Stokes Equations

The Parabolized Navier-Stokes (PNS) equations correspond to the steady state form of the TLNS equations. Streamwise diffusion is the only physical process neglected and the corresponding terms are dropped from the FNS equations along with the unsteady terms. Space marching techniques are employed to solve the PNS equations and the problem is well posed provided that the inviscid region of the flow is supersonic and that the streamwise velocity is positive everywhere. When the inviscid region is subsonic the stationary equations become elliptic and upstream influence occurs in the streamwise direction. In this case, numerical instabilities can arise and must be suppressed. Several methods have been developed to alleviate these difficulties (eg. Anderson et al., 1984) to extend the range of applicability of the PNS equations for subsonic flows. Although the solution of the PNS equations presents a large computational savings over solving the FNS equations, they are rarely used for the solution of turbomachinery flows due to their inability to handle streamwise flow separation. They are more commonly used for internal duct flows and are capable of dealing with cross flow separation and strong secondary flow (Hänel, 1991).

Other modifications can be made to the governing equations for solving special cases. The Partially Parabolized Navier-Stokes (PPNS) equations are similar to the PNS equations but the streamwise pressure gradient term is modified for subsonic regions to account for the upstream propagation of pressure waves. The pressure is treated as elliptic and stored for the whole domain. This enables the calculations of weak streamwise separation. The solution of these equations are substantially cheaper than the FNS equations but slightly more expensive than the PNS equations.

6.1.2 Discretization of the Governing Equations

The governing equations described in the previous section are discretized to replace the partial differential equations (PDEs) by a set of algebraic equations which can be solved numerically. A number of discretization methods are available, the most common being finite-difference, finite-volume, and finite-element methods.

6.1.2.1 Finite-Difference Approach

The finite-difference method is based on the discretization of the PDEs (for example, Equation 6.1) by using Taylor series expansions. For physical domains of arbitrary shape, the governing equations are first transformed to a generalized curvilinear coordinate system. The transformation is accomplished by mapping the arbitrary grid system in physical space (x,y,z) to a uniform, rectangular grid in computational space (ξ,η,ζ) . The generalized coordinate system can be expressed in several ways. One such method is the so-called body-fitted coordinate system, where for example, the two coordinate lines ξ and ζ are aligned along a surface in streamwise and transverse directions and where the third coordinate, η , is normal to the surface.

Transformation of the original PDEs complicates the original equations considerably. Transformation derivatives (or metrics) and the Jacobians of transformation must be calculated and incorporated into the governing equations. The metrics of transformation (for example, $\partial\xi/\partial x$) represent the ratio of arc lengths in computational space to those in physical space. The transformation Jacobians can be interpreted as the ratio of the volumes in physical space to those in computational space (Hoffman, 1989).

6.1.2.2 Finite-Volume Approach

Whereas the finite-difference approach is based on the differential formulation of the conservation equations (Equation 6.1), the finite-volume approach is based on the more fundamental integral formulation. Integration of Equation 6.1 over the volume of a cell leads to the equivalent integral relation:

$$\frac{\partial}{\partial t} \oint_V U dV + \oint (E_i + E_o) dA_i + \oint (F_i + F_o) dA_j + \oint (G_i + G_o) dA_k = 0 \quad (6.2)$$

The physical domain is divided into a number of adjacent finite volumes or cells which exchange mass, momentum, and energy. Hence, the formulation remains very close to the physics of the problem and is conceptually more straightforward than both finite-difference and finite-element methods. The main advantages of the finite-volume formulation are good conservation properties and direct applicability to physical space (x,y,z) (Fletcher, 1988a). Consequently, the finite-

volume formulation is less computationally expensive, but not necessarily more accurate, than the finite-difference formulation. The Dawes code, used in the present study, is based on the finite-volume discretization of the Thin-Layer Navier-Stokes equations. Further details are given in the following chapter.

6.1.2.3 Finite Element Approach

Finite-element methods were first developed for stress and displacement calculations in solid mechanics but have recently gained popularity in computational fluid dynamics. In the finite-element approach the domain is subdivided into finite elements and the solution vector $\mathbf{U}$ is interpolated using shape functions. A special formulation of the governing equations known as the weak formulation is used. This formulation employs weighting functions, W , that multiply the residual. The weak formulation of Equation 6.1 is defined as follows:

$$\oint_V \frac{\partial U}{\partial t} W dV + \oint_V \frac{\partial (E_i + E_v)}{\partial x} W dV + \oint_V \frac{\partial (F_i + F_v)}{\partial y} W dV + \oint_V \frac{\partial (G_i + G_v)}{\partial z} W dV = 0 \quad (6.3)$$

In this formulation, if the weighting functions are unity then the finite-volume formulation is recovered.

Nakajima and Kallinderis (1994) compared the finite-element approach to the finite-volume approach for several incompressible, viscous flow test cases. They found that the two methods were equally accurate with uniform grids but the finite-element method was less sensitive to grid nonuniformity. The two methods required approximately the same amount of computing time per cell per step. However, the finite-element approach required more than double the memory storage capacity. Dawes (1986b) rejected the finite-element approach on the grounds of complexity. Furthermore, the code was developed as a design tool intended for the modest computers of the mid 80's which had severe memory storage constraints.

6.1.3 Discretization Errors and Numerical Dissipation

The difference between the solution to a partial differential equation and its discretized counterpart is known as the *truncation (or discretization) error* which is either dissipative or dispersive. First-order accurate methods are associated with dissipation errors. These errors tend to decrease the gradients within the solution domain in the same way as physical dissipation (or viscosity). Second-order accurate methods are associated with dispersion errors. These errors cause oscillations in the solution in regions with high gradients.

Most Navier-Stokes solvers use central differencing which is primarily associated with dispersion error. It is often necessary to add artificial dissipation which is useful for attenuating oscillations in the solution. Oscillations can occur in the inviscid solution as high frequency oscillations and also more vigorously in the vicinity of shocks. Two types of artificial dissipation are generally employed. For the inviscid regions, randomly distributed small errors (eg. round-off errors) accumulate during the iteration process. These errors are distributed with the smallest resolvable wavelengths on the order of the local grid step sizes. Fourth-order differences, which are of $O(\Delta x)^3$, are used as damping terms to filter out high frequency oscillations in the inviscid regions but in viscous regions they may suppress small-scale vortical structures (Hänel, 1991). The fourth-order differences fail in regions of shock waves and are replaced by more vigorous second-order differences. The second-order differences, $O(\Delta x)$, are activated by a "shock sensor" in regions experiencing high pressure gradients. The terms may be inadvertently activated in boundary-layer regions thus impairing the viscous solution. Numerical dissipation, whether caused by truncation error or explicitly added to prevent oscillatory solutions, can impair the accuracy of predictions if it becomes the same order as the physical dissipation. More advanced artificial viscosity schemes counteract this problem and minimize the numerical dissipation effects in the viscous layers.

The Dawes code (Dawes, 1988) uses central differencing in its discretization scheme, which is second-order accurate. As mentioned above, second-order central differencing will lead to dispersion errors which are characterized by oscillatory solutions. Dawes adds artificial dissipation terms to the solution to attenuate the oscillations. Consequently, the added artificial dissipation may impair accurate loss predictions by adding to the physical dissipation. The artificial

Dissipation scheme used by Dawes is based on the method of Jameson and Baker (1983,1984). The details of the method are presented in the following chapter.

6.1.4 Methods of Solution

The time-dependent Navier-Stokes equations are commonly used for solving both transient and steady flow problems. The time integration can either be based on explicit methods or implicit methods.

6.1.4.1 Explicit Methods

The explicit schemes are the easiest to implement since the solution at each grid point is decoupled from all other grid points at the advanced time step. The main disadvantage of explicit methods is that the numerical time step is restricted by the Courant, Friedrichs, and Lewy (CFL) stability limit which states that the domain of dependence of the numerical scheme must contain the domain of dependence of the original equations (McNally and Sockol, 1985). The CFL number is usually a ratio of the time step to the cell step size normalized by some characteristic velocity. For viscous flows, the time step size is restrictively small since a close mesh spacing is required to resolve the boundary layers, particularly for high Reynolds number flows. However, this problem can be reduced for steady-state solutions by adjusting the time step everywhere to be near the CFL limit, thus providing a significant decrease in overall computation time. Explicit schemes are particularly suitable for predicting unsteady flows provided that they are second-order accurate in time, because the small time step imposed by the CFL stability limit is accommodated by the need for a small time step to predict the time-dependent solution. The most popular explicit schemes used for the Navier-Stokes equations are the multi-step Runge-Kutta methods (eg. Arnone and Swanson, 1993) and methods based on the predictor-corrector scheme of MacCormack (1969).

6.1.4.2 Implicit Methods

Implicit schemes are characterized by temporal backward differencing. The non-linear terms at the

advanced time level are linearized about their values at the previous time level (eg. Anderson *et al.*, 1984). Implicit schemes are computationally much more expensive than explicit schemes since they require the inversion of large solution matrices. For two-dimensional problems, the solution matrix is block pentadiagonal and for three-dimensional problems it is block septadiagonal. The equations are substantially simplified when the solution matrices are approximately factored to give a set of block tridiagonal matrices (eg. Hoffmann, 1989). The main advantage of implicit schemes is that they have much less restrictive stability properties than explicit schemes. This feature makes implicit schemes particularly suitable for the solution of steady-state flows. The most popular implicit schemes for the solution of the Navier-Stokes equations are those based on the methods of Briley and McDonald (1977) and Beam and Warming (1978).

From the previous discussion it is clear that implicit schemes and explicit schemes both have merits as well as drawbacks. Consequently, hybrid methods have been developed to take advantage of the merits of both schemes. One such method developed by MacCormack (1982), uses an explicit-implicit predictor-corrector approach which solves block bidiagonal equations, thus reducing computation time. Dawes (1988) uses a hybrid scheme which is based on a pre-processed Beam-Warming algorithm. The advantage of Dawes' algorithm is that the stability requirements are less restrictive than for explicit schemes. Furthermore, Dawes' algorithm is less computationally expensive than the usual Beam-Warming algorithm. A detailed discussion of the algorithm used by Dawes is given in the following chapter.

6.1.5 Turbulence Modelling

6.1.5.1 General Background

The time dependent Navier-Stokes equations are considered to govern turbulent flows in the continuum regime and solving these equations for the turbulence quantities is known as Direct Numerical Simulation (DNS). Unfortunately, DNS is not economically feasible at present since the time and space scales of turbulent motion would require restrictively small control volumes and time steps for their resolution. Large Eddy Simulation (LES) solves the time dependent Navier-Stokes equations for the evolution of large eddies (which are largely responsible for momentum transport) but models the smallest, subgrid eddies. Although LES is more

computationally efficient then DNS it is unlikely that it could be used as an economic engineering tool for quite some time (Karniadakis and Orszag, 1993). Lumley (1989) presents a good discussion on the potentials and limitations of DNS and LES.

Researchers have turned their attention to the development of turbulence models which involve the solution of a modelled form of the statistically averaged Navier-Stokes equations, the Reynolds equations. The Reynolds equations govern the mean time motion of the fluid and are identical to the laminar Navier-Stokes equations with extra apparent stress terms caused by the fluctuating components of velocity. These apparent stresses are commonly referred to as Reynolds stresses and are given by

$$(\overline{\tau_{ij}})_{turb} = -\rho \overline{u_i' u_j'} \quad (6.4)$$

Similar terms arise from the Reynolds energy equation and are known as Reynolds heat-flux terms. The Reynolds equations cannot be solved directly since they include the correlations of fluctuating quantities as additional unknowns. Additional equations involving the new unknowns must be found or assumptions must be made regarding the relationship between the time-mean flow variables and the new turbulent quantities. Consequently, turbulence models are needed for the closure of the Reynolds equations. Several excellent review papers have been written summarizing the variety of available turbulence models and their relative merits: of particular interest are the papers by Rodi (1982), Haines (1982), Marvin (1983) and Hieu (1992).

6.1.5.2 Mixing-Length Models

Mixing-length or algebraic models make use of the Boussinesq assumption which relates the Reynolds stresses to the rate of mean strain through an apparent scalar turbulent or "eddy" viscosity. The Boussinesq assumption is given by

$$-\rho \overline{u_i' u_j'} = \mu_T \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \left(\mu_T \frac{\partial u_k}{\partial x_k} + \rho \bar{k} \right) \quad (6.5)$$

where μ_T is the turbulent viscosity, δ_{ij} is the Kronecker delta function ($\delta_{ij}=1$ if $i=j$ and $\delta_{ij}=0$ if $i \neq j$), and the kinetic energy of turbulence is $\bar{k} = \overline{u_i' u_i'} / 2$.

Mixing-length models are phenomenological in nature: by analogy to the kinetic theory of gases, the turbulent viscosity is related to the characteristic turbulent velocity (u_T) and length (l) scales by

$$\mu_T = \rho u_T l .$$

Prandtl used $l|\partial u/\partial y|$ as the characteristic velocity which leads to

$$\mu_T = \rho l^2 \left| \frac{\partial u}{\partial y} \right| .$$

Hence, μ_T can be specified by determining the mixing-length distribution. The mixing-length can be thought of as the size of the dominant eddy which affects the local flow. Several methods exist for determining the mixing-length distribution; the most popular being the improved Cebeci-Smith model described by Nituch *et al.* (1978) or the model of Baldwin and Lomax (1978) which has gained popularity for internal flows.

Mixing-length models are sometimes referred to as local equilibrium models because they are implicitly based on the assumption that the turbulence is in local equilibrium, which means that turbulence energy is dissipated at the same rate as it is produced at every point in the flow. This is one of the major shortcomings of mixing-length models because they cannot account for the transport and history effects of turbulence. The limited empirical input used in these models results in a lack of universality. Although these models are suitable for analyzing shear-layers they begin to fail in complicated flows such as those involving large separations or three-dimensional flows in general. Despite all of these limitations, mixing-length models (particularly the model of Baldwin and Lomax) are still used very widely with considerable success.

In the present study, the Dawes code makes use of the Baldwin-Lomax turbulence model. Dawes (1986b) simplifies the original Baldwin-Lomax (1978) model but uses most of the constants as given in the original paper. The details of the turbulence model used by Dawes are presented in the following chapter. Further examples of mixing-length models for turbomachinery cases are given in Section 6.1.5.5 and performance comparisons are made to higher order models.

6.1.5.3 Eddy Viscosity Transport Models

In an effort to take transport and history effects of turbulence into account, models based on the transport of turbulence quantities have been developed. In this way, turbulence quantities are allowed to vary throughout the flow and are described by additional partial differential equations. The number of supplementary PDEs describes the class of model; for example, if one extra PDE is used then it is known as a one equation turbulence model. Eddy viscosity transport models still use the Boussinesq assumption and solve for the turbulence viscosity by using empirical relations between μ_T and the turbulence quantities. The most widely used of these models is the two-equation k- ϵ model. Two additional partial differential equations are solved: one for the turbulence dissipation rate, ϵ , and the other for the kinetic energy of turbulence, k . The turbulence viscosity is then given by

$$\mu_T = \frac{C_\mu \rho (k)^2}{\epsilon}$$

The k- ϵ model contains 5 empirical constants and it was hoped that constants could be found which would be applicable to all turbulent flows; however this has not been the case. Another limitation of the model is that it assumes isotropic eddy viscosity, which is not always observed experimentally. In recent years several new two equation models have been developed which appear to out perform the k- ϵ model (Wilcox, 1993). These include the k- ω and k- τ models (Speziale et al., 1992).

6.1.5.4 Stress Transport Models

Due to the lack of universality of the eddy viscosity transport models, a considerable amount of effort has been undertaken in the development of Reynolds stress models. Unlike the algebraic or differential eddy viscosity models mentioned above, the Reynolds stress models solve transport equations for the actual Reynolds stresses. Such models require the solution of three to five additional transport PDE's and contain 20 to 25 empirical constants. Due to the large number of constants, it is difficult to have a physical feeling for the meaning of the terms in the equations.

Reynolds stress models are in their infancy and are mainly used for research rather than as engineering tools. At the present time, they have not been tested for a wide range of complex flows and the universality of the approach is still uncertain. Furthermore, these models are very expensive since they require the solution of up to five additional PDE's.

A variant of Reynolds stress models known as algebraic stress models solve algebraic equations for the Reynolds stresses rather than differential equations. Consequently, they are more economic to solve but require a great deal of empiricism.

6.1.5.5 Turbulence Models for Internal Flows

The most popular turbulence models for internal flows are the Baldwin-Lomax mixing-length model and the two equation $k-\epsilon$ model. Turner and Jennions (1993) compared the Baldwin-Lomax model to the $k-\epsilon$ model for the modelling of flow through transonic fans. They found that the Baldwin-Lomax model predicted pressure rises from shocks to be too far aft, but improvements were attained when new constants were employed as suggested by Granville (1987). They also concluded that shock positions and flow rates were predicted better and convergence was faster with the $k-\epsilon$ method.

Cleak and Gregory-Smith (1992) compared a very simple mixing-length model to the $k-\epsilon$ model for predicting secondary flows in a turbine cascade. They found that the mixing-length model best predicted the mean flow properties: specifically predicting greater vortex strength and mobility and lower losses, whereas the $k-\epsilon$ model best predicted the turbulence quantities. These results seem to favour the mixing-length model over the $k-\epsilon$ model, in contradiction to the findings of Turner and Jennions.

Dawes (1990) compared predictions in turbomachinery blade rows using an algebraic mixing-length model and a one equation model for turbulent kinetic energy. The higher order model was expected to be more accurate since it takes upstream history effects into account, and while being more expensive, is more physical. Dawes found that although the one equation model allowed better predictions of separation bubbles, the loss predictions of both models were of comparable

performance. The reason for this is that 90% of the entropy generation in turbulent boundary layers occurs deep within the boundary layer at y^+ values around 10-30. Both turbulence models prescribe the same eddy viscosity in the inner layers, where the losses themselves are being generated.

As mentioned in Section 6.1.5.2, the code used in the present study (Dawes' BTOB3D) uses the Baldwin-Lomax model. It is clear from the above discussion that the Baldwin-Lomax model is competitive if not better than the more advanced $k-\epsilon$ model in terms of accurate prediction of losses. Loss prediction is one of the main points of interest in this study.

6.1.6 Grid Generation

The numerical solution of the governing partial differential equations requires discretization of the PDE's to obtain a set of algebraic equations which are solved at a set of discrete points in the domain of interest. Determination of the location of the discrete points in the domain and on the boundaries is known as grid generation.

The distribution of grid points is very important for the quality of solution in terms of numerical accuracy and stability. The primary requirements of the grid are accurate representation of geometric boundaries, clustering of grid points in regions of rapid changes, smooth distribution of step sizes, and orthogonality. Generally, all requirements cannot be satisfied and compromises have to be made.

Grid density is another very important issue. Dorney and Davis (1991) suggest that researchers will not be able to separate the effects of turbulence modelling on flow solutions from those caused by truncation error and numerical dissipation until they can affordably and reliably produce grid independent solutions. Very large grid densities necessary for accurately resolving viscous flow physics and convection of vorticity, require computational resources far beyond the means of most engineering facilities.

Grid generation is a highly developed research area with dozens of papers being written on the

topic; for an excellent review, see Thompson (1984) and Thompson *et al.* (1982). A brief summary of the main methods used in turbomachinery applications is presented below.

6.1.6.1 Structured Grids

Meshes with grid points, regularly distributed in some order are known as structured grids. Structured grids are the most commonly used for today's turbomachinery applications and can support all methods of discretization. The main drawbacks to structured grids are that mesh singularities can arise and adaptive gridding is difficult.

Typical examples of structured mesh configurations commonly used in turbomachinery applications are **C**-grids, **O**-grids and **H**-grids. **O**-type grids indicate a coordinate system having lines encircling the body as shown in Figure 6.1a. **C**-type grids indicate a system with lines emanating from a boundary, passing around a body and returning to the boundary. Examples of **C**-grids are given by Davis *et al.* (1987) and Amone and Swanson (1993) and shown in Figure 6.1b. **H**-grids are perhaps the most commonly used configuration used in turbomachinery design codes (eg. Dawes, 1988) due to their simplicity and economy. The particular **H**-grid configuration employed by Dawes for the code used in the present study is presented in Chapter 7. Turner *et al.* (1993) point out that **H**-grids sometimes become inaccurate in regions of high skew and aspect ratio. They recommend a modified grid that is nearly orthogonal which they termed an **I**-grid and found that it performed better than the standard **H**-grid. As will be shown in Chapter 8, the **H**-grid used by Dawes code appeared to give satisfactory results.

Not only is the grid type important for establishing accurate results but also the details of the mesh configuration can have important effects. Chung and Baek (1992) investigated three variations of **C**-type grids and found that they had significant effects on the predictions in the trailing-edge regions of turbine cascades.

6.1.6.2 Solution-Adaptive Unstructured Grids

Unstructured grids consist of an unordered arrangement of grid cells and have recently been

Figure 6.1a: Example of an O-mesh. (Reproduced from Raw et al., 1989)

Figure 6.1b: Example of a C-mesh. (Reproduced from Arnone and Swanson, 1993)

gaining popularity as grid generators for finite-element and finite-volume methods. The most recent version of Dawes' code uses unstructured grids (Dawes, 1992 and 1993). Solution-adaptation implies that the grids are refined automatically as the solution progresses in regions that require higher resolution to capture the local flow phenomena accurately. The main advantages of unstructured grids are as follows:

- (i) Greater geometrical flexibility allowing for geometries of arbitrary complexity and high resolution in local regions without the necessity for global refinement (unlike structured methodologies).
- (ii) Well suited for the implementation of grid adaption (as a function of both time and space) in the form of grid enrichment or depletion in areas experiencing strong gradients or high truncation errors; this allows for user independent solutions since no *a priori* knowledge of the solution is required for the construction of the optimal mesh.
- (iii) High degrees of accuracy are possible by refining the mesh since small local elements will increase the local accuracy without the risk of developing badly deformed or skewed elements.

Unstructured grids are still in the development stage but have recently been implemented for turbomachinery analysis, for example: Dawes (1992,1993) and Siden *et al.* (1990). Most of the codes using unstructured grids are currently used in research environments and are not yet competitive with conventional structured techniques. Typically, to achieve comparable accuracy and detail takes about three times as long with the unstructured mesh than with the structured mesh due to increased computer storage requirements for the unstructured mesh. In general, a three-dimensional unstructured solution adaptive mesh is very complicated and can take far more cpu time to generate than the actual flow simulation (Dawes, 1992). An example of an unstructured grid for a turbine blade analysis is given in Figure 6.2 .

Figure 6.2: Example of an unstructured grid with solution-adaptive refinement in regions with high gradients. (Reproduced from Siden *et al.*, 1990)

6.2 Aerodynamics of Turbine Cascades

As mentioned in the introduction, the present study is concerned with the effects of flow convergence and divergence on the midspan performance characteristics of a cascade of turbine blades. Midspan losses and exit flow deviation were the main performance parameters investigated. Section 6.2.1 briefly describes the various sources of loss in turbine cascades and Section 6.2.2 briefly summarizes the investigations carried out to date on the effects of axial velocity density ratio on cascade flows.

6.2.1 Loss Mechanisms in Turbine Cascades

In general, losses can be divided into two groups; those whose mechanisms are based on two-dimensional effects and those whose mechanisms are based on three-dimensional effects. Losses in two-dimensional cascades are usually referred to as profile losses. These losses can be further broken down into components such as boundary-layer losses, trailing-edge losses and mixing losses. For high speed cascades, shock losses also play an important role. Three-dimensional effects also contribute to loss due to the presence of endwall boundary layers, secondary flows and tip-leakage flows.

6.2.1.1 Losses Due to Two-Dimensional Flow Phenomena

Boundary-layer losses arise due to friction between adjacent layers of fluid near the blade surface. For most boundary layers, the velocity changes most rapidly near the wall and thus entropy generation (or loss) is highest in the inner layer. As noted earlier, 90% of the entropy generation in turbulent boundary-layers occurs deep within the boundary layer at y^+ values around 10-30 (Dawes, 1990). In turbines, the blade suction surface typically experiences significant areas of laminar flow. Laminar boundary layers produce less loss due to their milder velocity gradients. Figure 6.3, from Denton (1993), shows that for Reynolds numbers where both laminar and turbulent boundary layers can exist, the dissipation, C_d (a measure of entropy production), in laminar boundary layers is 2 to 5 times less than in turbulent boundary layers. On the other hand, turbulent boundary layers have less tendency to separate in regions of adverse pressure gradient or in interactions with shock waves. This emphasizes the importance of predicting transition on

Figure 6.3: Dissipation coefficient for laminar and turbulent boundary layers. (Reproduced from Denton, 1993)

Figure 6.4: Individual components of loss, $Re_{ex}=1,000,000$. (Reproduced from Mee et al., 1992)

turbine blades. For turbine blades, Mee *et al.* (1992) found that boundary layer losses decrease monotonically with increasing Mach number but are the dominant source of loss at subsonic conditions, as shown in Figure 6.4 . Furthermore, they found that the losses are dominated by the suction side boundary layer since the pressure side boundary layer is an order of magnitude thinner.

Aside from boundary layers, additional mixing losses arise due to entropy creation whenever a fluid is subject to viscous shear. High rates of shearing often occur in the trailing-edge region of blade passages due to the differences in fluid velocity and pressure from the suction and pressure surfaces. This component of mixing loss is usually termed trailing-edge loss. The other component of mixing loss, termed wake loss, is less important and is due to the mixing out of the wake behind the trailing edge. Immediately behind the trailing edge is a region of low velocity fluid at nearly uniform pressure, the base pressure. It has been found that trailing edge losses rise with decreasing base pressure. Trailing-edge losses contribute at least 1/3 of the total losses in most Mach number regimes as seen in Figure 6.4 .

Flow passing through a shock-front experiences high viscous stresses and heat conduction leading to the creation of entropy. The resulting shock losses contribute about 1/3 of the total losses at Mach numbers near unity as seen in Figure 6.4 .

6.2.1.2 Losses Due to Three-Dimensional Flow Phenomena

Experimental tests require the use of finite span blades which typically have aspect ratios (span/chord) of 1-3. The formation of endwall boundary layers and secondary vortices have considerable influence on the overall loss. Sharma and Butler (1987) indicate that in turbine cascades, these losses, often termed endwall losses or secondary losses, can account for as much as 30-50 percent of the total loss.

The development of secondary flows in turbine cascades is a complicated process and has received considerable attention most notably by Langston *et al.* (1977) and more recently by Sieverding (1985a, 1985b). The strength of secondary flows depends on the thickness of the inlet boundary

2

PM-1 3½"x4" PHOTOGRAPHIC MICROCOPY TARGET
NBS 1010a ANSI/ISO #2 EQUIVALENT

Figure 6.5a: The formation of secondary flows according to Langston et al. (1977).

Figure 6.5b: The formation of secondary flows according to Sharma and Butler (1987).

layer and on the degree of turning in the blade row. Figure 6.5a illustrates the formation of secondary flows according to Langston *et al.* (1977) and Figure 6.5b illustrates a more recent model proposed by Sharma and Butler (1987).

Perdichizzi (1990) studied the effects of Mach number on the development of secondary flows in turbine cascades. He concluded that as transonic and supersonic regimes are approached, the secondary losses become relatively less important than profile losses primarily due to high losses associated with shock/boundary-layer interactions.

6.2.2 The Effects of Axial Velocity Density Ratio on Cascade Flows

As mentioned in the introduction, one possible measure of flow two-dimensionality is the axial velocity density ratio (AVDR) or simply axial velocity ratio (AVR) for incompressible flows. AVDR is a measure of the acceleration or deceleration of flow in a cascade. It is defined as:

$$AVDR = \frac{\int_0^s (\rho_2 V_{a2})_{ms} dy}{\int_0^s (\rho_1 V_{a1})_{ms} dy} , \quad (6.6)$$

where the subscripts 1, 2, and ms refer to leading-edge plane, trailing-edge plane and midspan respectively. Figure 6.6, reproduced from Rodger (1992), illustrates the three possible values of AVDR. A flow with an axial deceleration has an $AVDR < 1$, whereas a flow with an axial acceleration has an $AVDR > 1$, and a flow with constant axial velocity will have an $AVDR = 1$.

The effect of AVDR on midspan performance parameters has received considerable attention for compressor cascades. These include investigations by Heilmann (1972), Starken *et al.* (1975), Stark and Hoheisel (1981) and Starke (1981). An extensive literature review of the work done on compressor cascades was performed by Rodger (1992) and only a summary of his findings is presented here. It was generally found that an increase in AVDR led to decreased total pressure loss, a slight linear decrease in flow deviation, a thickening of the midspan boundary layer, and a shift of the blade static pressure curve to lower values, with reductions in suction side adverse

Figure 6.6: Possible values of the axial velocity ratio (AVR). (Reproduced from Rodger, 1992)

pressure gradients. Although the strength of the relationships varied from study to study, the trends were generally the same. A summary of the results (as reported by Rodger) is given in Table 6.1.

Very little data have been published regarding the effects of AVDR on the midspan performance parameters of turbine cascades. Kiock *et al.* (1986) tested the same turbine blade section in four different transonic facilities with different test sections and instrumentation. All facilities were equipped with parallel endwalls without boundary layer suction. They concluded that AVDR had little effect on midspan loss or outlet flow angle in the range of AVDR from 0.9 to 1.0, unlike in compressor cascades. Sharma and Graziani (1983) studied the influence of endwall secondary flows on the development of suction surface boundary layers at midspan of turbine cascades. They concluded that the AVDR effect, in itself, is insufficient to explain the influence of endwall flows on the aerodynamics and heat transfer behaviour on the suction surface. Unfortunately, both studies did not include the data to quantify their conclusions.

Rodger (1992) and Rodger *et al.* (1992) performed in house studies in an attempt to quantify the effects of AVDR on the midspan performance parameters of a low-speed cascade of turbine blades. Further work was undertaken by Whitehouse (1993) and Whitehouse *et al.* (1993) with a different family of turbine blades. It is more convenient to show these results in direct comparison to the results of the present investigation; therefore they are presented in Chapter 8.

The motivation of the present study was to extend the work of Rodger to transonic Mach numbers. The effects of flow convergence and divergence on total pressure losses, exit flow deviation, blade pressure distribution and base pressure were investigated for both design and off-design incidences. The results are presented in Chapter 8.

Table 6.1: Summary of Literature Survey from Rodger (1992)

	author	year	Theoretical Experimental	AVR control method	blade section investigated	AVR increase effect on Y_t	AVR increase effect on deviation	AVR increase effect on C_p	other parameters examined
1	Kubota	1962	T&E	none	compressor	.	.	.	incr i = incr AVR
2	Pollard et al.	1963	T	.	compressor	.	linear decr	lower values	.
3	Pollard et al.	1967	E	suction	compressor	.	linear decr	lower values	AR, Re, Tu
4	Stark	1972	T&E	suction	compressor	decr for +i incr for -i	linear decr (incr in deflection)	lower values	i, s/c
5	Heilmann	1972	T&E	suction	compressor	strong decr +i weak decr -i	linear decr (incr in turning)	lower values	i, M, Tu
6	Foltner	1972	T&E	suction	compressor	decr	decr (decr outlet angle)	.	M, i
7	Ohuonu et al.	1973	E	suction	compressor	.	linear decr	lower values	i, camber, stagger
8	Masek et al.	1973	E	suction	compressor	strong decr +i weak decr -i	linear decr (incr in deflection)	lower values	i, Tu
9	Ilyas et al.	1973	E	suction	compressor	decr AVR < 1.05 incr AVR > 1.05	decr (incr AVR > 1.1 at M=0.13)	lower values	M
10	Dunham	1974	T	.	compressor	.	.	.	b.l. vs AVR
11	Starken et al.	1975	E	suction	compressor	decr	decr (decr outlet angle)	lower values	M, i
12	Galapati et al.	1976	T&E	suction, contra- cting endwalls	compressor	.	decr	lower values	AR, s/c
13	Starke	1981	T&E	contracting endwalls	compressor	strong decr +i weak decr -i	linear decr (incr in turning)	lower values	i, b.l. thickness, diffusion ratio
14	Stark et al.	1981	T&E	suction	compressor	decr	linear decr (incr in turning)	lower values	AR, i, M
15	Sharma et al.	1983	E	.	compressor & turbine	.	.	.	b.l. behavior
16	Kiock et al.	1985	E	solid parallel walls	turbine	no effect AVR=0.9 to 1.0	no effect AVR=0.9 to 1.0	.	wind tunnel and instrumentation

Computational Model

7

A computational investigation of the effects of AVDR on the midspan performance parameters of a cascade of turbine blades has been conducted using a three-dimensional Navier-Stokes solver (BTOB3D) developed by Dawes. A number of papers have been published by Dawes (eg. 1985, 1986a, 1986b, 1986c, 1987, 1988) describing various versions of the code.

Section 7.1 presents a detailed description of the version of BTOB3D used in this study. The information has been gathered by consulting all of the above mentioned papers and cross-checking the equations with the actual BTOB3D source code. Shortcomings of the code have been identified and suggestions have been made for possible improvements.

Section 7.2 describes the geometry of the blade and the cascade investigated in the present study. The section concludes with a presentation of the mesh refinement study which was used to determine an appropriate mesh size and configuration necessary for gathering essentially grid-independent results. The actual results are presented in the Chapter 8.

7.1 Three-Dimensional Navier-Stokes Solver (BTOB3D)

Dawes developed BTOB3D as an analysis tool for turbomachinery flows. The code has been extensively tested for a wide range of turbomachinery cases including the transonic flow in a compressor rotor (Dawes, 1987), secondary flow in a transonic turbine cascade (Dawes, 1986a), the tip clearance flow in an axial compressor (Storer and Cumpsty, 1993), and the flow in a turbine cascade with trailing-edge damage (Sjolander *et al.*, 1993).

7.1.1 Governing Equations

The general Full Navier-Stokes equations have been presented in Section 6.1.1.1. The form of the equations used by Dawes in BTOB3D are presented below.

The three-dimensional, compressible Navier-Stokes equations are written in integral conservation form and cast in the blade relative frame using cylindrical coordinates (r, θ, x) as follows:

$$\frac{\partial}{\partial t} \oint_V U dV = \oint H dA + \oint \rho S dV \quad (7.1)$$

where $\mathbf{U}$ represents the flow variables, $\mathbf{H}$ represents the various fluxes, $\mathbf{S}$ contains the body forces, and A and V are the areas and volumes of the control volumes respectively. $\mathbf{U}$, $\mathbf{H}$, and $\mathbf{S}$ are given as:

$$\mathbf{U} = \begin{bmatrix} \rho \\ \rho V_x \\ r \rho V_\theta \\ \rho V_r \\ \rho E \end{bmatrix} \quad \mathbf{H} = \begin{bmatrix} \rho \bar{q} \\ \rho V_x \bar{q} + \bar{\tau} \hat{i}_x \\ r \rho V_\theta \bar{q} + r \bar{\tau} \hat{i}_\theta \\ \rho V_r \bar{q} + \bar{\tau} \hat{i}_r \\ \rho I \bar{q} \end{bmatrix} \quad \mathbf{S} = \begin{bmatrix} 0 \\ 0 \\ -2\Omega r V_r \\ \frac{V_\theta^2}{r} + r\Omega^2 + 2\Omega V_\theta \\ 0 \end{bmatrix}$$

with the relative velocity:

$$\bar{\mathbf{q}} = V_x \hat{i}_x + V_r \hat{i}_r + V_\theta \hat{i}_\theta$$

rotation speed:

$$\Omega$$

the rothalpy:

$$I = h_{0rel} - \frac{1}{2}(\Omega r)^2 = \gamma E - \frac{\gamma-1}{2} [\bar{\mathbf{q}} \cdot \bar{\mathbf{q}} - (\Omega r)^2]$$

and the stress tensor given by:
$$\bar{\tau} = \begin{vmatrix} -p + \tau_{xx} & \tau_{rx} & \tau_{\theta x} \\ \tau_{rx} & -p + \tau_{rr} & \tau_{r\theta} \\ \tau_{\theta x} & \tau_{r\theta} & -p + \tau_{\theta\theta} \end{vmatrix}.$$

As discussed in Section 6.1.1.2, Dawes uses the thin-layer approximation, thus neglecting viscous stress terms involving streamwise derivatives ($\partial/\partial x$). The stresses are then written:

$$\begin{aligned} \tau_{xx} &= -\frac{2}{3}\mu \left(\frac{\partial V_r}{\partial r} + \frac{1}{r} \frac{\partial V_\theta}{\partial \theta} \right), & \tau_{rx} &= \mu \frac{\partial V_x}{\partial r}, & \tau_{\theta x} &= \frac{\mu}{r} \frac{\partial V_x}{\partial \theta} \\ \tau_{rr} &= \frac{2}{3}\mu \left(2 \frac{\partial V_r}{\partial r} - \frac{1}{r} \frac{\partial V_\theta}{\partial \theta} \right), & \tau_{rx} &= \tau_{rx}, & \tau_{r\theta} &= \mu \left(\frac{1}{r} \frac{\partial V_r}{\partial \theta} - \frac{V_\theta}{r} + \frac{\partial V_\theta}{\partial r} \right) \\ \tau_{\theta\theta} &= \frac{2}{3}\mu \left(\frac{2}{r} \frac{\partial V_\theta}{\partial \theta} + 3 \frac{V_r}{r} - \frac{\partial V_r}{\partial r} \right), & \tau_{x\theta} &= \tau_{\theta x}, & \tau_{r\theta} &= \tau_{r\theta}. \end{aligned}$$

The above system (Equation 7.1) represents five equations which are in order: continuity, x-momentum, θ -momentum, r-momentum, and energy. To close the system, the equation of state, expressed in terms of the total internal energy (E), is used:

$$p = \rho(\gamma - 1) \left(E - \frac{1}{2} [\bar{q} \cdot \bar{q} - (\Omega r)^2] \right)$$

The Baldwin-Lomax (1978) mixing-length model is employed to model the turbulence. The Baldwin-Lomax model is outlined in detail in Section 7.1.5.

As seen from Equation 7.1, Dawes simplifies the energy equation by eliminating the viscous stress and heat conduction terms. Essentially, the energy equation is being replaced by a statement of conservation of rothalpy. This is a reasonable assumption for high speed flows with no external heat addition, although the range of validity may be exceeded in flows with large temperature variations (eg. Fletcher, 1988b, Cumpsty, 1989, and Lyman, 1993). Furthermore, the laminar viscosity is assumed constant in the entire flow domain and is based on the Reynolds number

entered as an input to BTOB3D (see Appendix 2A). For the present study, the simplifications to the energy equation and the laminar viscosity calculation are reasonable and should not effect the predictions to any great extent.

7.1.1.1 Compressibility Considerations

As mentioned above, the code solves the *compressible* Navier-Stokes equations and is not strictly suitable for incompressible calculations for three reasons. Firstly, the ever diminishing density variations become comparable to the round-off error and would require extra precision to be resolved. Secondly, a restrictively high number of time steps would be needed since the CFL stability criterion (defined in Section 6.1.3) is related to the speed of sound such that the time step approaches zero as the speed of sound approaches its incompressible value of infinity. Lastly, the long-wavelength errors present in the solution have difficulty interacting with the inflow and outflow boundaries as the Mach number decreases. For example, an error wave running downstream has its magnitude modified proportional to $(1-M)/(1+M)$ after one reflection from the outflow boundary and one reflection from the inflow boundary (Dawes, 1990). Clearly, for small Mach numbers the errors decay very slowly. In a more recent version of the code, Walker and Dawes (1990) have introduced artificial compressibility to the compressible flow equations using Chorin's method (Chorin, 1967). By adding artificial compressibility, the code is able to investigate flows at low, subsonic Mach numbers where compressibility effects are negligible.

Although the version of the code used in this study is intended for compressible calculations, it has been applied approximately to incompressible flows by using Mach numbers which are as low as the code will allow.

7.1.2 Computational Mesh

Of the possible grid configurations discussed in the previous chapter, Dawes prefers the **H**-mesh (see Figure 7.1) due to its simplicity and flexibility. The mesh consists of three families of surfaces: streamwise (x,θ), quasi-orthogonal (r,θ), and meridional (x,r) as seen in Figure 7.2. The spacing between elements need not be uniform and are specified by the user. Dawes (1985) recommends that the relative spacing between elements should not exceed $\pm 30\%$ to maintain

Figure 7.1: The configuration and notation of the H-mesh.

Figure 7.2: Illustration of the three families of surfaces found in the H-mesh.

acceptable accuracy. The only geometrical limitation on the grid is that the streamwise and quasi-orthogonal surfaces are surfaces of revolution.

The resulting mesh consists of a three-dimensional network of hexahedral cells. The geometry is defined by the coordinates of the cell corners or nodes, whereas the flow variables themselves are stored at cell centres. In a finite-volume formulation it is necessary to find the volume of each cell and the areas of all its faces. Each cell has six faces; two bladewise, two streamwise and two quasi-orthogonal as shown in Figure 7.3. The areas are represented by A_B , A_S , and A_Q respectively. The cells can be arranged in a general orientation and hence each face will in general have three components of area in each of the three coordinate directions (r, θ, x). Since each face is defined by four arbitrary vertices it is in general not planar. The approximate area of a particular face is obtained from a vector product involving the two diagonals (Vinokur, 1989). For example, if the position vectors of the vertices of a bladewise face are given by r_1 , r_2 , r_3 , and r_4 (see Figure 7.4), then the area is approximated by

$$A_B = \frac{1}{2} (r_1 - r_2) \times (r_3 - r_4) = \frac{1}{2} \begin{vmatrix} \hat{i}_r & \hat{i}_\theta & \hat{i}_x \\ (r_1 - r_2) \hat{i}_r & (r_1 - r_2) \hat{i}_\theta & (r_1 - r_2) \hat{i}_x \\ (r_3 - r_4) \hat{i}_r & (r_3 - r_4) \hat{i}_\theta & (r_3 - r_4) \hat{i}_x \end{vmatrix}$$

which yields the three components of area of the bladewise face

$$A_B = A_{Br} \hat{i}_r + A_{B\theta} \hat{i}_\theta + A_{Bx} \hat{i}_x \quad (7.2)$$

where A_{Br} represents the r -component of area of the bladewise face, $A_{B\theta}$ represents the θ -component of area of the bladewise face, and A_{Bx} represents the x -component of area of the bladewise face. The area components for streamwise and quasi-orthogonal faces are calculated in a similar manner; however, the components $A_{S\theta}$ and $A_{Q\theta}$ are zero since the streamwise and quasi-orthogonal surfaces are surfaces of revolution. In other words, the θ -component of area of the bladewise face, $A_{B\theta}$, is constant for all values of θ (or i) for a given j, k location. Consequently, the volume of each cell can be calculated by multiplying $A_{B\theta}$ of the cell by the average of the lengths of the four sides (of constant j, k location) of each cell as follows:

Figure 7.3: Illustration of the three different faces for the hexahedral cell.

Figure 7.4: Calculation of area for the bladewise face of a hexahedral cell.

$$V = \frac{A_{pe}}{4} [(r\theta_5 - r\theta_1) + (r\theta_7 - r\theta_3) + (r\theta_6 - r\theta_2) + (r\theta_8 - r\theta_4)] . \quad (7.3)$$

7.1.3 Discretization of the Governing Equations

7.1.3.1 Finite-Volume Formulation

The governing equations are expressed in integral conservation form (Equation 7.1) and are discretized using the finite-volume formulation, as described in Chapter 6. The flow variables (U) are stored at the centres (ijk) of the hexahedral cells. Each integral in Equation 7.1 is replaced by a discrete summation around the faces of the hexahedral cells as follows:

$$\frac{\Delta U_{ijk}}{\Delta t} \Delta V_{ijk} = \sum_{Cell(ijk)} H \Delta A + \rho S_{ijk} \Delta V_{ijk} + D_{ijk} = R_{ijk} \quad (7.4)$$

where $R_{ijk} = (\Delta U / \Delta t) \Delta V_{ijk}$ is the residual at cell (ijk).

The fluxes are calculated by summing the contributions through all six cell faces of every cell. These fluxes are determined by linearly interpolating values between adjacent cell centres and then multiplying the projection of area normal to that direction; hence the spatial accuracy of the method is second-order. Smoothly varying grids, having relative spacing of no more than $\pm 30\%$, are required to maintain the accuracy and to ensure close global conservation. Since the viscous stress terms still contain first derivatives, a local curvilinear coordinate system must be defined for the determination of these terms. The present version of the code uses a mesh transformation which is only first-order accurate. The experience of Dawes and others is that this should not cause a problem provided that the spacing between successive grid points does not differ by more than $\pm 30\%$ (Sjolander, 1994). The additional term (D_{ijk}) in Equation 7.4 is an adaptive artificial viscosity term which will be further discussed in Section 7.1.3.3 .

7.1.3.2 Solution Algorithm

The algorithm time marches the unsteady Navier Stokes equations in finite-volume form to a steady solution using a two-step explicit/one-step implicit method. The algorithm is similar to the Beam-Warming (1978) or Briley-MacDonald (1977) implicit schemes but premultiplies the algorithm by a preprocessing matrix. The idea of the preprocessing matrix is based on the relaxation solution of the Euler equations by Johnson (1982) and is used because the centered difference convection terms in the usual Beam-Warming algorithm are associated with a lack of diagonal dominance in the coefficient matrix for large time steps and high Reynolds numbers. By premultiplying the basic Beam-Warming algorithm by an appropriate preprocessing matrix and by eliminating terms that cannot be supported on a three point molecule to ensure a tridiagonal structure, diagonal dominance is guaranteed for any time step size (Dawes, 1986b). The equations are represented in the so-called delta form with the two explicit steps:

$$\Delta U^{(1)} = \frac{\Delta t}{\Delta V} R_{jk}^n; \quad U^{(1)} = U^n + \Delta U^{(1)} \quad (7.5)$$

$$\Delta U^{(2)} = \frac{\Delta t}{\Delta V} R_{jk}^{(1)} \quad (7.6)$$

followed by the implicit equations which are approximately factored to give:

$$\left[I - \epsilon_i \frac{\Delta t^2}{\Delta V^2} \lambda_i^2 \delta_{ii}^2 \right] \cdot \left[I - \epsilon_j \frac{\Delta t^2}{\Delta V^2} \lambda_j^2 \delta_{jj}^2 \right] \cdot \left[I - \epsilon_k \frac{\Delta t^2}{\Delta V^2} \lambda_k^2 \delta_{kk}^2 \right] \Delta U = \Delta U^{(2)} \quad (7.7)$$

$$U^{n+1} = U^n + \Delta U \quad (7.8)$$

where the superscripts n, 1, n+1 are present, intermediate and next time steps respectively. ϵ_i , ϵ_j , and ϵ_k are free parameters, and λ_i , λ_j , and λ_k are the spectral radii of the flux Jacobians in the i, j, and k directions (Fletcher, 1988b). Spectral radii can be used instead of the complete eigenvalues (Rieger and Jameson, 1988) to simplify the algorithm by requiring the solution of a tridiagonal matrix rather than a more complicated block tridiagonal matrix. The traditional central difference operator is defined as

$$\delta_{ii}^2(\psi) = \psi_{i+1} - 2\psi_i + \psi_{i-1}.$$

For simplicity and computational efficiency Dawes chose

$$\lambda_i = \lambda_j = \lambda_k = \frac{\Delta V_{ijk}}{\Delta t_{ijk}}.$$

Premultiplication of the LHS matrix of Equation 7.7 does not disrupt the formal accuracy of the converged solution since only steady solutions are sought. As steady state is approached, ΔU (LHS of Equation 7.7) approaches zero; this is the advantage of using the delta form. Consequently, alterations to the LHS matrix of Equations 7.7 do not affect the accuracy, but merely provide time marching stability. Due to the pre-multiplication, a two-step explicit procedure (Equations 7.5-7.6) is required for the solution of the RHS of Equation 7.7.

The algorithm is stable for any value of the CFL number provided that the free parameters (ϵ) are chosen large enough, but extensive testing by Dawes has shown that optimum convergence is achieved for CFL values in the range of 1 - 3 and free parameters set to about unity (Dawes, 1988).

The algorithm is very similar to a two-step Runge-Kutta scheme with implicit residual smoothing (eg. Jameson and Baker, 1983). Jameson and Baker have shown that the restriction on the time step imposed by the CFL condition is eliminated by utilizing implicit residual smoothing. Stability is ensured provided that the stability parameter (analogous to the free parameter in Equation 7.7) is large enough.

7.1.3.3 Artificial Viscosity

Numerical dissipation (and dispersion) is introduced into the solution due to truncation errors associated with representing the governing equations with discretized approximations. These errors become a problem when the magnitude of the numerical dissipation is of the same order as the physical dissipation. However, dissipation (numerical or physical) effectively attenuates short wavelengths while having little effect on long wavelengths (Fletcher, 1988b). This feature

can be exploited by adding artificial dissipation (or viscosity) to high Reynolds number flows or flows with discontinuities to attenuate short wavelength errors.

Short wavelength errors arise when using central space differencing in convection-dominated flows. These errors arise due to odd-even point decoupling of solutions and oscillations in the vicinity of shock waves. Dawes adopts an artificial viscosity method based on that developed by Jameson and Baker (1983,1984) for the Euler equations.

The dissipation operator consists of three terms which represent the contributions from each of the three spatial directions

$$D(U) = D_I + D_J + D_K \quad (7.9)$$

where for example

$$D_I = d_{i+1/2,j,k} - d_{i-1/2,j,k}$$

The right hand side terms have the form

$$d_{i+1/2,j,k} = FT \frac{\Delta V}{\Delta t} \left[\epsilon_{i+1/2,j,k}^{(2)} (U_{i+1,j,k} - U_{i,j,k}) - \epsilon_{i+1/2,j,k}^{(4)} (U_{i+2,j,k} - 3U_{i+1,j,k} + 3U_{i,j,k} - U_{i-1,j,k}) \right]$$

where $FT = (\Delta t / \Delta s) \cdot a_0$ and a_0 is the absolute stagnation speed of sound at the inlet.

To inhibit odd-even point decoupling of the solutions a fourth-difference dissipation term is used. To permit shock capture without oscillations a more vigorous second-difference term is used which is of order one in the vicinity of shocks. To maintain the second-order accuracy of the solution away from the immediate vicinity of shocks, a shock sensing switch is used. The coefficients $\epsilon^{(2)}$ and $\epsilon^{(4)}$ are calculated in a self adaptive manner as follows:

$$\epsilon_{i+1/2,j,k}^{(2)} = \text{the minimum of} \left[\frac{1}{2}, \frac{K^{(2)}}{2} (\pi_{i,j,k} + \pi_{i+1,j,k}) \right],$$

$$\epsilon_{i+1/2,j,k}^{(4)} = \text{the maximum of} \left[0, K^{(4)} - \frac{\alpha}{2} (\pi_{i,j,k} + \pi_{i+1,j,k}) \right]$$

where the shock sensor is sensitive to second-differences in the pressure;

$$\pi_{i,j,k} = \frac{|p_{i+1,j,k} - 2p_{i,j,k} + p_{i-1,j,k}|}{p_{i+1,j,k} + 2p_{i,j,k} + p_{i-1,j,k}}.$$

The values of the constants recommended by Dawes are $K^{(2)}=1$, $K^{(4)}=0.01$, and $\alpha=2$. The artificial viscosity is set to zero in directions normal to solid boundaries for cells adjacent to the boundaries to avoid masking the physical viscosity. Note that the above equations are not exactly the same as those presented by Dawes (1985, 1986b, and 1987) but are consistent with the expressions which appear in the code presently used at Carleton University.

Swanson and Turkel (1987) improved the basic model of Jameson by incorporating anisotropic dissipation terms to account for high aspect ratio cells associated with high Reynolds number flows to reduce the artificial dissipation in the streamwise direction. Van der Burg *et al.* (1992) proposed a modification to Jameson's scheme for improved shock capturing. This is done by replacing the central difference scheme by an upwind scheme which takes the characteristic directions more into account, making the results more accurate near shocks.

7.1.3.4 Boundary Conditions

Several boundary conditions must be imposed on the computational domain, including those at inflow, outflow and periodic boundaries as well as on solid boundaries.

According to the theory of characteristics, three quantities must be specified along a subsonic inflow boundary (Arnone and Swanson, 1993). In the present code, the total temperature, total pressure, and flow angle are fixed. The flow angle consists of two components: the pitch angle is usually set to zero to denote zero radial component of velocity, and the swirl angle is set to

adjust the incidence of the inlet flow. These conditions are fixed at the first cell centres ($j=1$). Zero static pressure gradient is selected in the streamwise direction between the first two grid points ($p_{1ik}=p_{2ik}$) so that all of the remaining variables can be determined. The upstream boundary must be placed sufficiently far upstream of the leading-edge plane, where no viscous phenomena occur, so that the total pressure is known and fixed. From experience, a distance of about half an axial chord upstream of the blades is sufficient (Dawes, 1986b).

For a subsonic axial outlet, the average static pressure corresponding to the desired isentropic exit Mach number is specified. If the blade passage is an annular sector then the outlet pressure is specified at the hub and the radial distribution is determined from radial equilibrium, as follows:

$$\frac{\partial p}{\partial r} = \rho \frac{V_\theta^2}{r} \quad (7.10)$$

All other variables are found by linear extrapolation as follows:

$$\psi_{i,jM-1,k} = 2\psi_{i,jM-2,k} - \psi_{i,jM-3,k} \quad (7.11)$$

where $jM-1$ is the index of the last cell centre. For supersonic axial outlets, all of the flow variables are extrapolated.

The mesh is constructed such that cell faces are aligned with the walls and along the periodic boundaries. Hence, the cells adjacent to periodic boundaries are treated as if they were interior cells and the flux across periodic boundaries is calculated by linear interpolation between cells at either side of the boundary.

Boundary conditions on solid walls (hub, casing, and blades) are no slip and no flux of mass, momentum and energy, resulting in a specification of no velocity on the surface. Wall static pressure is found by setting the gradients of pressure normal to the wall to zero, which is a reasonable assumption for high Reynolds number flows. The wall pressure is specified by

$$p_w = \frac{3}{2}p_{1jk} - \frac{1}{2}p_{2jk} \quad (7.12)$$

The known velocity at the cell center adjacent to the wall and the known zero velocity at the wall

along with a skin friction correlation based on the log-law, is used to determine the wall shear stress, τ_w . The skin-friction coefficient is given by

$$C_f = -0.001767 + \frac{0.03177}{\log^{-1}(Re)} + \frac{0.25614}{[\log^{-1}(Re)]^2} \quad (7.13)$$

where the Reynolds number is based on local conditions and normal distance to the wall. The skin-friction coefficient is related to the wall shear stress by

$$C_f = \frac{\tau_w}{\frac{1}{2} \rho \bar{q} \cdot \bar{q}} \quad (7.14)$$

The wall shear stress is used in the three momentum equations by resolving it into its three components and multiplying by the area of the cell face representing the wall as follows:

$$\begin{aligned} \sum F_w &= \tau_w A_w \hat{i}_x + \tau_w A_w \hat{i}_r + \tau_w A_w \hat{i}_\theta \\ &= \tau_w A_w \frac{V_x}{\sqrt{q_w \cdot q_w}} \hat{i}_x + \tau_w A_w \frac{V_r}{\sqrt{q_w \cdot q_w}} \hat{i}_r + \tau_w A_w \frac{V_\theta}{\sqrt{q_w \cdot q_w}} \hat{i}_\theta. \end{aligned}$$

The log-law is a well known approximation for the variation of velocity in a two-dimensional, incompressible, turbulent boundary layer experiencing zero pressure gradient. The log-law is generally valid in the range $10 < y^+ < 500$, where

$$y^+ = \frac{y \sqrt{\tau_{wall} / \rho}}{\nu}.$$

More complicated functions have been published allowing for free-stream pressure gradients and compressibility effects (Patankar and Spalding, 1972). Consequently, it is necessary to resolve the boundary layer into the log-law region in order to get reasonably accurate predictions for the wall shear stress. In other words, the centers of the computational cells adjacent to solid boundaries should be in the applicable range of the log-law.

For inviscid endwalls, the wall shear stress is set to zero. This option is used to simulate planes of symmetry or to achieve effectively two-dimensional calculations. Unfortunately, this option does not appear to be implemented properly in the present version of the code. A shear stress on the inviscid endwalls is predicted and the flow tangency condition does not seem to be satisfied. The problem appears to originate in the formulation of the viscous stress terms and how they are modified for the inviscid endwall option. Furthermore, the turbulence model must be modified when using the inviscid endwall option as will be further discussed in Section 7.1.5.

7.1.4 Multigrid Acceleration

Multigrid methods were developed to maximize the convergence of numerical solutions by following the premise as stated by Brandt (1982): "The amount of computational work should be proportional to the amount of real physical changes in the computed system". Stalling of the numerical process can arise in time marching problems when the time steps (dictated by numerical stability requirements) are much smaller than the time scales associated with the changes in the solution or when the spatial and temporal scales of the grid in certain parts of the computational domain are too fine compared to the physical changes to provide efficient convergence. These problems arise due to the "stiffness" of the problem: different components to the solution are associated with drastically different time and length scales. For example, physical and numerical instabilities initially develop as low-frequency and high-frequency modes respectively. The high-frequency errors have scales on the order of the fine grid and are efficiently attenuated; however coarser meshes are required to transport the low-frequency errors to the far-field boundaries. Consequently, multiple grid levels lead to a reduction in time to steady state convergence. This is the so called Full Approximate Storage Scheme (FAS) methodology developed by Brandt (1977) and it has been adopted by Dawes.

Dawes uses one level of multigrid which he claims will roughly halve c.p.u. time for a given solution. The process involves sweeps that transfer residuals from the fine grid to the coarse grid and then interpolate corrections from the coarse grid to the fine grid. In this way, each level attenuates errors whose wavelengths are comparable to that level's mesh size resulting in efficient reduction of all errors. The multigrid process is initiated from the outset and cycles back and

forth between fine and coarse grids until final steady state convergence is achieved.

In outline, the following multigrid strategy is used (similar to Jameson and Yoon, 1986). Coarser meshes are obtained by eliminating every second grid line in each coordinate direction. Consequently, one coarse cell consists of eight fine cells. On the coarse grid, the solution is initialized as volume weighted averages so that conservation is maintained:

$$U_{2h}^n = \frac{\sum U_h V_h}{\sum V_h} = \frac{\sum U_h V_h}{V_{2h}} \quad (7.15)$$

where the subscripts h and $2h$ refer to fine and coarse cells respectively. For the coarse grid solution, a forcing function, P , is added to the governing equations. The forcing function is calculated from the fine grid solutions as follows:

$$P = \sum \Delta U_h V_h / \Delta t_h = \sum R_h \quad (7.16)$$

where ΔU_h is determined from Equation 7.7. The explicit steps are solved on the auxiliary grid in an analogous way to Equation 7.5 and Equation 7.6 respectively, as follows:

$$\text{step 1:} \quad U_{2h}^{(1)} = U_{2h}^n + \frac{\Delta t_{2h}}{V_{2h}} P \quad (7.17)$$

$$\text{step 2:} \quad \Delta U_{2h}^{(2)} = \frac{\Delta t_{2h}}{V_{2h}} (R_{2h}^{(1)} + P) \quad (7.18)$$

The auxiliary grid solver then solves Equation 7.7 for ΔU_{2h} . The corrections are then interpolated from the coarse mesh to the fine mesh as follows:

$$U_h^{n+1} = U_h^{n+1} + I_{2h}^h \Delta U_{2h} \quad (7.19)$$

where I_{2h}^h is a linear interpolation function that transfers the coarse grid solutions to the fine grid

The artificial dissipation model for the fine grid is replaced by a simple scaled second order model for the auxiliary grid. Furthermore, the viscous stress terms are not calculated on the auxiliary grid.

7.1.5 The Baldwin - Lomax Turbulence Model

As discussed in Chapter 6, a wide variety of models exist for the modelling of turbulence. Dawes favours the simple mixing-length models due to their simplicity and computational efficiency. Turbomachinery flows are largely governed by the effects of blockage and any turbulence model which predicts a sufficient amount of blockage in the correct areas should be successful. Most mixing-length models possess this feature. Furthermore, as mentioned in the previous chapter, Dawes (1990) has found that loss predictions using mixing-length models are of comparable performance to higher order models.

The Baldwin-Lomax (1978) mixing-length model has gained wide acceptance for predictions involving internal flows. Other mixing-length models such as that developed by Cebeci and Smith (Sjolander, 1993) require a well defined boundary layer with identifiable outer edge to determine the length and velocity scales. For turbomachinery flows, it is difficult to clearly identify the edge of the boundary layer, especially since the free stream is often a shear flow. The Baldwin-Lomax model determines the length scales from the vorticity distribution in the boundary layer thus alleviating the need to define an outer edge.

The Baldwin-Lomax model is a two layer model in which the eddy viscosity is determined from

$$\mu_t = \text{the minimum of} \begin{bmatrix} (\mu_t)_{inner} \\ (\mu_t)_{outer} \end{bmatrix}. \quad (7.20)$$

The Prandtl mixing-length formulation is used in the inner region

$$(\mu_t)_{inner} = \rho l^2 |\omega| \quad (7.21)$$

where the mixing-length is given by

$$l = ky.$$

The usual Van-Driest damping function, $1 - e^{-y^+/A^+}$ is omitted thus eliminating the need to calculate y^+ . The damping function only becomes significant very close to the wall ($y^+ < 50$) and can be neglected for the typical mesh densities used. The magnitude of the vorticity is given by

$$|\omega| = \sqrt{\left(\frac{\partial u}{\partial y} - \frac{\partial v}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial z} - \frac{\partial w}{\partial y}\right)^2 + \left(\frac{\partial w}{\partial x} - \frac{\partial u}{\partial z}\right)^2}.$$

For the outer region, which is used in wakes as well as in attached and separated boundary layers, the eddy viscosity is given by

$$(\mu_t)_{outer} = K C_{cp} \rho F_{wake} F_{Kleb} \quad (7.22)$$

where

$$F_{wake} = y_{max} F_{max}.$$

The quantities F_{max} and y_{max} are determined from the function

$$F(y) = y |\omega|$$

where F_{max} is the maximum value of $F(y)$ in a given profile and y_{max} is the value of y at which the maximum occurs. The Van-Driest damping function is again omitted from the equation for $F(y)$, which is the usual procedure in wakes. The Klebanoff intermittency factor is given by

$$F_{Kleb} = \left\{ 1 + 5.5 \left(\frac{C_{Kleb} y}{y_{max}} \right)^8 \right\}^{-1}.$$

The various constants used by Dawes in the current code are the same as those published by Baldwin and Lomax for constant pressure boundary layers at transonic speeds.

$$\begin{array}{ll} A^* &= 26 \\ KC_{cp} &= 0.0268 \end{array} \quad \begin{array}{ll} C_{Kleb} &= 0.3 \\ k &= 0.41 \end{array}$$

To apply the Baldwin-Lomax formulation to three-dimensions, Dawes finds the distance normal to each solid boundary (hub, casing, or blade) for each node in the blade passage and determines the eddy viscosity for a given node based on the length scale associated with the nearest wall. Upstream of the blade passage, only the hub and casing are considered for determination of the length scale. Downstream of the blade passage (in the wakes), the outer formulation (Equation 7.22) is used and the terms F_{max} and y_{max} are determined from length scales based on the blade surfaces in the trailing-edge plane. An oversight regarding the turbulence model is present for the

inviscid endwall option. For this case, the model incorrectly considers the endwalls for determination of the length scales. To remedy this problem, the turbulence model should only consider the blade surfaces in the blade passage. Upstream of the blade passage, the model should be switched off entirely, since, away from the (viscous) endwalls, the viscosity is essentially the laminar value. No modification is necessary downstream of the blade row.

Many investigations have suggested that the original Baldwin-Lomax model is not strictly suitable for predicting complex flows involving regions of cross-flow separation or shock/boundary-layer interaction. Visbal and Knight (1984) tested the Baldwin-Lomax model for two-dimensional supersonic compression corner interactions. They found that the constants C_{cp} and C_{Kleb} were dependent on Mach number and that the outer function was not suitable for the determination of the length scale in the separation regions investigated. Visbal and Knight proposed modifications to remedy these problems. However, the models still failed to predict the rapid recovery of the boundary layer downstream of reattachment.

Granville (1987) found that for boundary layers in pressure gradients (favourable or adverse) the constant values of C_{cp} and C_{Kleb} were unsatisfactory for satisfying the velocity similarity laws. Granville proposed formulas for the variation of C_{Kleb} with a modified Clauser pressure gradient parameter and for the variation of C_{cp} with C_{Kleb} .

The code (BTOB3D) was demonstrated at the 1994 ASME-IGTI Turbo Expo and it was found that the code did not accurately predict the wake development with downstream distance. Woods, from NASA-Lewis, concluded that wake predictions were improved when the original Baldwin-Lomax constant C_{cp} was reduced from 1.6 to 1.2 (Sjolander, 1994).

Dorney and Davis (1991) used the Baldwin-Lomax model for analyzing a low speed, linear turbine cascade. They found that the original turbulence model did not work well in corner flows such as the blade/endwall juncture. Due to the ambiguity of determining the mixing-length associated with the nearest wall, the predicted eddy viscosity in the corner regions can become discontinuous across constant $i=k$ (see Figure 7.1) lines which can cause large gradients to occur in the flow field. Dorney and Davis introduced a blending function to remove the discontinuity. The Dawes code does not use a similar blending function which may result in poor predictions

in blade/endwall corner regions. This problem is not a concern in the present study. For future studies, the blending function proposed by Dorney and Davis can easily be implemented into the version of Davies' code used at Carleton University.

Although the Baldwin-Lomax turbulence model suffers from a few shortfalls (as do all other conventional turbulence models), it should be adequate for giving reasonable predictions of profile loss for the cases of interest in this study.

7.2 Simulation of Cascade Flow

7.2.1 Cascade Geometry

Experimental studies on the effects of lack of flow two-dimensionality on turbine cascade performance were conducted by Rodger (1992) in a low speed, linear cascade test section. The blade section used in that study was taken from the midspan section of a power turbine blade from a midsize Pratt & Whitney Canada turboprop engine. The same blade section was used in the present study to validate the code against Rodger's results and to serve as an extension of Rodger's work to transonic conditions. The cascade geometry is given in Figure 7.5 and the coordinates of the blade surfaces are presented in Appendix 2B.

Rodger developed adjustable end-plates to manipulate the flow acceleration through the blade passage. These end-plates were originally designed to restart the endwall boundary layer and thus thin it, but the addition of convergence or divergence of the end-plates also allowed the axial velocity ratio through the cascade to be varied. A flexible knife-edge leading edge was used to guide the flow onto the end-plates without a separation bubble.

In the computational model, endwall convergence or divergence was employed to control the flow acceleration or deceleration. The planar endwalls were symmetrically angled (about midspan) from the inflow to the outflow boundary. The endwall boundary layer grows from the inflow boundary, starting from a prescribed total pressure distribution. In the present study, uniform total

Inlet flow angle α_1	28.35°	Inlet metal angle β_1	29.29°
Exit flow angle α_2	57.65°	Exit metal angle β_2	57.79°
Design incidence i_{des}	-0.94°	Design deviation δ_{des}	-0.14°
Stagger angle γ	23.06°	Camber angle	87.08°
Chord length C	162.81 mm	Pitch/Chord ratio S/C	0.68
Axial chord length C_r	149.27 mm	Aspect ratio H/C	1.23
Pitch length S	110.71 mm	Wedge angle (l.e.)	52.4°
Blade span H	244.22 mm	t_{max}/C	0.182
Leading-edge diameter	9.43 mm	Trailing-edge thickness	4.2 mm
Maximum thickness	29.63 mm		

Figure 7.5: Blade and cascade geometry.

pressure profiles were specified at the inlet. Therefore, the inlet boundary layers developed from an initial thickness of zero.

Rodger's experimental setup used blades with an aspect ratio of 1.22 without the end-plates and about 1 with the end-plates. For the computational model, all final runs were performed for an inlet aspect ratio of 1.5. Aspect ratios of unity were investigated but it was found that the span was insufficient for the development of a substantial region of uniform loss at midspan. Conversely, large aspect ratios were undesirable since stretching the span would lead to cells of high aspect ratio unless more cells were added in the spanwise direction.

As mentioned in Section 7.1.1, the code was developed in cylindrical coordinates. Consequently, the blade passages are modelled as annular sections rather than as linear sections. To approximately model the blade passage as a linear section, it was necessary to use a very large hub radius. This minimizes the radial pressure gradients resulting from the swirling flow in an annulus. Tests were run at a hub radius of about 8.8 metres (ie. 500 blades around the annulus) and re-run for a ten-fold increase (88 metres or 5000 blades). It was found that the radial change in pressure along the span was reduced by one order of magnitude: from about 1000 Pa to less than 100 Pa. Also, the resulting spanwise distribution of loss coefficients appeared to be essentially symmetric about midspan for the higher blade count test. Thus, 5000 blades were used in all of the final runs.

7.2.2 Mach Number Considerations

As mentioned in Section 7.1.1.1, the Dawes code was developed for compressible flows and is not usable for incompressible predictions. The code is often compared with incompressible results, (the present study included) for which case it is preferable to run the code at the lowest Mach numbers possible.

Yaras (1988) attempted to run the code for an inlet Mach number (M_{in}) of 0.1 and found that the solution was unstable and did not converge. The lowest Mach number for which stable solutions were achieved by Yaras was for an inlet Mach number of 0.29. Isaacs (1993) also achieved stable solutions for an inlet Mach number, M_{in} , of 0.29; the corresponding outlet Mach number, M_{out} , was

Figure 7.6: Mass-averaged total pressure loss vs. trailing-edge Mach number. (Reproduced from Isaacs, 1993).

0.41. Although the solutions were stable and converged to a low RMS residue (defined in the next section), the predictions for $M_{\infty} < 0.6$ were found to produce unrealistic losses, as seen in Figure 7.6. Similarly, in the present study, stable solutions were achieved for exit Mach numbers of about 0.45, but the predictions of total pressure loss were unrealistic.

In light of the above observations, care must be taken whenever low Mach number predictions are being performed. Reducing residuals is a necessary but not sufficient requirement to guarantee realistic predictions. For low speed predictions, it is recommended that conservatively high Mach numbers should be used to avoid erroneous predictions. In the present study, the subsonic results were obtained for exit Mach numbers of about 0.6. These results are presented in the following chapter.

7.2.3 Convergence Criteria

The Dawes code has no internal criterion which determines the achievement of convergence to the steady-state solution. The user must select the number of time steps to be run by the code. It is important that a sufficient number of time steps is selected to ensure convergence. The number of time steps necessary for convergence is linked to the Mach number and the fineness of the grid through the CFL number (defined in Section 6.1.4.1). Generally, for higher Mach numbers and/or for finer grids, more time steps are necessary to achieve convergence.

The Dawes code supplies information regarding convergence history in two output files (see Appendix 2A). Typical trends in several parameters are presented in Figure 7.7 for a transonic case with parallel endwalls. One quantity used to judge convergence is the RMS residue. The RMS residue is a measure of the imbalance in mass, momentum, and energy for every cell in the computational mesh. As the steady-state solution is approached, the RMS residue converges to some low value [typically $O(10^{-5})$] as seen in Figure 7.7a. The oscillations in the RMS residue suggest that convergence to "machine zero" has been achieved by about 3000 time steps.

RMS residue is not always a reliable quantity for identifying converged solutions but it is useful for indicating whether the solution is stable or unstable. Several Dawes code results were

Figure 7.7a: RMS residue vs. time steps.

Figure 7.7b: Maximum % error of mass flow through quasi-orthogonal planes.

Figure 7.7c: % Error of mass flow through selected quasi-orthogonal planes.

presented at the 1994 ASME-IGTI Turbo Expo (Sjolander, 1994). There was a general consensus that RMS residue was not the best quantity to indicate convergence. Many participants judged convergence on the basis of the mass flow rate through several quasi-orthogonal reference planes (see Section 7.1.2) along the flow. Figure 7.7b shows the evolution of mass flow rate (in terms of maximum per cent error) with time step. The mass flow rate through each of the quasi-orthogonal planes is referenced to the inlet plane and the plane with the maximum error is recorded. Figure 7.7c shows the evolution of mass flow rate through selected planes (in this case: leading-edge plane, trailing-edge plane, and exit plane) referenced to the mass flow rate through the inlet plane. From Figure 7.7c, it can be seen that the mass flow rate through the leading-edge plane levels off after about 1500 time steps, whereas for the exit plane the mass flow rate levels off after about 3500 time steps. Figure 7.7c also shows that the error in mass flow through the leading-edge plane converges to a higher value than for the other planes. This may result from the difficulty of the H-mesh to deal with the leading edge of the blade as will be evident from Mach number contours shown in the next section.

Although the above mentioned quantities are useful for indicating the approximate time step required for convergence, the most reliable quantities are the flow variables themselves. Dawes (1988) notes that the loss coefficient is the last parameter to converge. Tests have shown that for the case shown in Figure 7.7, the loss coefficient achieved essentially a steady-state value after 4000 time steps. It is always good practice to be generous with the number of time steps allowed.

7.2.4 Selection and Refinement of the Computational Mesh

7.2.4.1 Selection of the Computational Domain

As mentioned in Section 7.1.3.4, Dawes (1986b) has found that it is usually sufficient to place the inlet boundary about one half of an axial chord length upstream of the leading-edge plane. The outflow boundary must be placed at an adequate distance downstream of the trailing-edge plane so that flow distortions, such as secondary vortices and wakes, sufficiently mix out. Tests have shown that extending the domain one axial chord length downstream of the trailing-edge plane is sufficient for reliable solutions.

7.2.4.2 Refinement of the Blade-to-Blade Mesh

A mesh refinement study was performed to evaluate the grid density in the blade-to-blade plane required to establish essentially grid-independent solutions (not considering three-dimensional effects). Grid-independence is achieved when no improvement in results is gained with additional mesh refinement. The mesh refinement study was performed for Mach numbers high enough to avoid problems with incompressibility. Also, the main goal of the present work was to extend Rodger's experimental results to transonic Mach numbers. Thus it was sensible to perform the mesh refinement study at higher Mach numbers. The Mach numbers for the test runs are quoted in terms of exit isentropic Mach number (M_{2s}) determined by:

$$\frac{p_{01}}{p_2} = \left[1 + \frac{\gamma - 1}{2} M_{2s}^2 \right]^{\frac{\gamma}{\gamma - 1}} \quad (7.23)$$

where p_{01} and p_2 are the inlet and exit boundary conditions respectively. In these terms, the mesh refinement study was performed at $M_{2s} = 0.98$.

Several grid densities were investigated and are shown in Figure 7.8. The memory constraints imposed by the computer used in the present study (Sun Sparc Station II with 32 Mbytes of RAM) allowed modelling a maximum of roughly 130,000 nodes using 32-bit real numbers. All blade-to-blade refinement tests were performed with equal numbers of nodes in the spanwise direction (k), namely 33. The blade-to-blade meshes investigated were as follows (pitchwise $\times$ axial $\times$ spanwise):

Grid 1:	17 $\times$ 35 $\times$ 33
Grid 2:	21 $\times$ 61 $\times$ 33
Grid 3:	27 $\times$ 89 $\times$ 33
Grid 4:	33 $\times$ 107 $\times$ 33

Each increase represents an approximate doubling of the number of cells in the blade-to-blade mesh. In terms of boundary-layer resolution on the blade surfaces, the minimum distance from the blade surface to the first cell centres on the pressure side and the corresponding y^+ values were as follows:

(a) Grid 1: 17×35

(b) Grid 2: 21×61

(c) Grid 3: 27×89

(d) Grid 4: 33×107

Figure 7.8: Blade-to-blade mesh configuration for grids 1, 2, 3, and 4.

(a) Grid 1: 17×35

(b) Grid 2: 21×61

(c) Grid 3: 27×89

(d) Grid 4: 33×107

Figure 7.9: Midspan mach number contours for grids 1, 2, 3, and 4.

Grid 1:	$y/c = 3.92 \times 10^{-3}$,	$y^+ \approx 190$
Grid 2:	$y/c = 3.00 \times 10^{-3}$,	$y^+ \approx 150$
Grid 3:	$y/c = 2.20 \times 10^{-3}$,	$y^+ \approx 120$
Grid 4:	$y/c = 1.74 \times 10^{-3}$,	$y^+ \approx 100$

Considerably denser meshes would be required to resolve the boundary layer down to the viscous sublayer ($y^+ < 10$). Dorney and Davis (1991) found that 900,000 nodes were required to model a single turbine blade row in order to obtain a $y^+ \approx 0.35$ and to establish grid-independence. This order of mesh refinement is currently not possible in most engineering facilities. Of more pertinence, is the number of nodes sufficient to produce results with acceptable accuracy.

As previously mentioned, the number of time steps required to obtain steady-state convergence is dependent on the fineness of the mesh. Denser meshes require longer run times. The approximate number of time steps required to obtain convergence for the following grids are:

Grid 1:	1000 time steps
Grid 2:	1500 time steps
Grid 3:	2500 time steps
Grid 4:	3000 time steps

To study the dependence of the solutions on the grid densities it is useful to investigate total pressure loss coefficients and Mach number contours.

Midspan Mach number contours are shown in Figure 7.9 . It can be seen that for grids 1 and 2, the contours show a considerable degree of oscillation. Also, the shock near the trailing edge on the suction side is smeared over many cells and the wakes are not properly resolved. Grids 3 and 4 show crisper shocks and wakes as well as fewer oscillations.

Total pressure loss coefficients (Y') at midspan are compared in Figure 7.10. For the course grids (grids 1 and 2), the losses are predicted to be substantially higher than for the fine grids (grids 3 and 4). For grids 3 and 4 the losses are predicted to be within $2\frac{1}{2}\%$ of one another. Based on these results, it would appear that the solution is reaching some level of grid-independence for grids 3 and 4. However, the presence of oscillations near the leading and trailing edges is still a concern and may be due to insufficient clustering in these regions.

Figure 7.10: Pitchwise mass-averaged loss coefficient, Y' , at midspan for different mesh configurations.

A new clustering procedure was developed to better resolve the leading and trailing edges while using the same number of total nodes. It was expected that the new grid would produce results with fewer oscillations and resolve the flow more accurately in the leading and trailing-edge regions. Both clustering algorithms are described in Appendix 2C. The grid generated by the new clustering algorithm, grid 5, has $33 \times 103 \times 33$ nodes and converged in 4000 time steps. Figure 7.11 shows the comparison between the blade-to-blade meshes and the midspan Mach number contours between grids 4 and 5, while Figure 7.12 shows the differences in leading and trailing-edge resolution. Grid 5 shows significantly fewer oscillations over grid 4 in the vicinity of the leading edge and on both pressure and suction surfaces. Also, the wake is seen to be much crisper, which can be attributed to the trailing-edge grid refinement. As can be seen in Figure 7.10, the total pressure loss coefficients (Y') decreased by about 8% from grid 4 results to grid 5 results. The reduction in loss can likely be attributed to an overprediction of trailing-edge losses by grids 1 through 4 due to the poor trailing-edge grid refinement. Further grid refinements may yield further small changes in loss, particularly with regard to trailing-edge loss. Due to limited computing capability, it was not possible to investigate increased refinements. Given the results predicted by the present mesh (see Chapter 8), confidence in predicted trends if not absolute values seems justifiable. As discussed in Chapter 6, the accuracy of predicting absolute loss is not only limited by grid configuration and refinement but also by other factors such as numerically generated dissipation and turbulence modelling.

A final note regarding the blade-to-blade mesh concerns its orientation. As seen in Figure 7.8, the mesh is aligned with the blade exit metal angle. The participants at the 1994 ASME-IGTI Turbo Expo concluded that extending the mesh in the approximate flow direction was necessary for proper wake resolution (Sjolander, 1994). By extending the downstream grid in the axial direction the wakes quickly moved to coarse grid regions, which resulted in poor shear flow predictions.

7.2.4.3 Refinement of the Meridional Mesh

Aside from the blade-to-blade mesh, the clustering of the meridional mesh in the spanwise direction was also an important consideration. Clustering the spanwise nodes near the endwalls

Figure 7.11a: Computational mesh and midspan mach number contours for grid 4.

Figure 7.11b: Computational mesh and midspan mach number contours for grid 5.

Figure 7.12a: Close-up view of the leading-edge region of grid 4 and grid 5.

Figure 7.12b: Close-up view of the trailing-edge region of grid 4 and grid 5.

(Figure 7.13a) is necessary to resolve the endwall boundary layers. Resolving these boundary layers, particularly upstream of the blade row, effectively increased the size and strength of secondary flows predicted by the code. 33 spanwise nodes are insufficient to accurately resolve these secondary flows. The presence of the passage vortex does not introduce an appreciable amount of blockage since the axial velocity in the vortex is essentially the same as the free-stream value (Sharma and Graziani, 1983). Therefore, there is no need to resolve the secondary flows, particularly since the main interest of this study is to investigate the performance parameters of the blade at midspan. In addition, endwall clustering caused two problems (shown in Figure 7.14): firstly, regions of physically unrealistic negative loss were predicted near the endwalls; and secondly, non-uniform results were predicted at midspan. The non-uniformities at midspan were particularly troublesome in light of the nature of the present study. Evidently, poor results at midspan were caused by the coarseness (and high aspect ratio) of the cells at midspan. A likely cause of the negative losses near the endwalls is the lack of a smoothing term in the Baldwin-Lomax model (1978) for the blade/endwall regions, as discussed in Section 7.1.5.

Uniform spanwise node spacing (shown in Figure 7.13b) alleviated the problem of negative losses near the endwalls and reduced the size and strength of the predicted secondary flow. However, problems of asymmetry arose and were particularly noticeable for extreme AVDRs, as shown in Figure 7.14 for $AVDR=1.1$. It is difficult to determine the cause of the asymmetry. One might suspect that it was caused by the finite hub radius effect (see Section 7.2.1) and that spanwise pressure gradients were still present. This effect can be dismissed because no clear pattern was discerned in the asymmetry. For example, in one case where 41 nodes were used in the spanwise direction the total pressure losses were asymmetrically higher near the casing, whereas for another case where 33 nodes were used in the spanwise direction the total pressure losses were asymmetrically higher near the hub. The asymmetry typically diffused into the midspan region resulting in erroneous non-uniform midspan predictions. The asymmetry may have been caused by a lack of grid-independence near the endwalls resulting in solutions that converged to erroneous values.

To remedy the asymmetry problem midspan clustering was tried (see Figure 7.13c). Not only would midspan clustering reduce the resolution of endwall effects and secondary flows, but it would also provide the finest grids at midspan, which was the primary region of interest. The

Figure 7.13a: Meridional mesh with endwall clustering.

Figure 7.13b: Meridional mesh with no clustering.

Figure 7.13c: Meridional mesh with midspan clustering.

Figure 7.14a: Pitchwise mass-averaged loss coefficient, Y' , at $x/C_t=1.4$ for different meridional mesh configurations.

Figure 7.14b: Expanded scale of Figure 7.14a.

effect of midspan spanwise clustering was to reduce the asymmetry to negligible levels and to reduce the strength of the secondary flows as well as the endwall boundary layers, as seen in Figure 7.14. The only adverse effect of midspan clustering was to diffuse the secondary flows further away from the endwalls towards midspan. This was due to the large cells close to the endwalls. However, this problem was not a concern since there was still a sufficiently large region (30% of span) of uniform losses at midspan. An attempt to remedy this problem was made by using a fine cell adjacent to the endwalls followed by coarse cells in the outer regions. The new clustering arrangement was essentially the same distribution as shown in Figure 7.13c, but the outermost coarse cells were replaced by fine cells. By doing this, one would expect that the boundary layers would be computed to be thinner and would also move the secondary vortices closer to the endwalls. The losses near the endwalls increased but the endwall effects were not further removed from the midspan region. However, the midspan predictions were nearly identical, indicating reasonable midspan grid-independence of the spanwise clustering arrangement.

A sample file for a transonic case with parallel endwalls using the grid 5 blade-to-blade mesh configuration and midspan spanwise clustering is given at the end of Appendix 2C.

Presentation and Discussion of Results

8

This chapter presents the computational results for the investigation of the effects of axial velocity-density ratio on the midspan performance parameters of the CC2 turbine cascade. Comparisons between measurements (obtained from Rodger, 1992) and computations for low speed flow are presented for design and off-design cases in Section 8.1, to assess the performance of the Dawes code. Further results are presented in Section 8.2 for two transonic speeds, at design and off-design incidences. Finally, Section 8.3 compares the results for the three levels of Mach number for design conditions, indicating the relative importance of the AVDR effect in different Mach number regimes.

8.1 The Effects of AVDR for Subsonic Conditions

The experimental investigation, conducted by Rodger (1992), was performed on a low speed cascade at Mach numbers around 0.1 . Such low Mach numbers cannot be simulated by the code, for reasons discussed in Section 7.2.2. Therefore, the subsonic computational test cases were calculated at an exit isentropic Mach number of about 0.6. The deviations between measured and predicted results due to Mach number effects will be discussed where appropriate. The influence of AVDR on aerodynamic parameters, such as profile loss, exit flow deviation, base pressure, and blade loading will be presented for design and off-design cases.

8.1.1 Design Incidence

8.1.1.1 Profile Loss

The profile loss coefficient used in the present study is based on the loss of total pressure and is defined as follows:

$$Y = \frac{P_{o1} - P_{o2}}{P_{o2} - P_2} .$$

The results will be presented in terms of a pitchwise mass-averaged loss coefficient, which is defined as follows:

$$Y' = \frac{1}{m} \int_0^s Y dm .$$

The loss coefficient results have been extracted from a plane located 40% of an axial chord length downstream of the trailing-edge plane, as was done by Rodger (1992). A small amount of mixing loss may occur downstream of this location but tests have shown that this additional loss is negligible. Goobie (1989) and Rodger (1992) both calculated a 4% value for mixing loss for the same measurement plane. The additional mixing loss has not been added to the computational results since trends are of greater interest than absolute values.

The computational results are presented with the measured results at midspan in Figure 8.1. The

Figure 8.1: The effect of AVDR on midspan profile loss (Y'_{me}) at design incidence for computational and experimental, subsonic results.

Figure 8.2: The effect of AVDR on exit flow deviation (δ_{me}) at design incidence for computational and experimental, subsonic results.

agreement with experiment is poor, both in terms of trend and absolute value. The computational results indicate a mild, linear increase in profile loss with AVDR; whereas, the experimental results indicate that loss is higher and more sensitive to values of AVDR less than unity. For values of AVDR above unity, the losses are relatively constant.

As mentioned in Section 6.2.2, an increase in AVDR results in flow acceleration through the blade passage. Denton and Cumpsty (1987) have shown that the loss production in the blade boundary layers is roughly proportional to the cube of the local freestream velocity. Consequently, the profile losses would be expected to rise with increasing AVDR. The experimental results indicate higher losses for $AVDR < 1$. Rodger attributed this trend to higher trailing-edge losses, as indicated by the lower base pressures, shown in Figure 8.4.

The profile loss coefficient is overpredicted by the Dawes code for values of AVDR above unity, as seen in Figure 8.1. It may be suspected that the discrepancy in Mach number between computations and experiments would affect the accuracy of prediction. However, from Figure 6.4 it appears that the loss is largely insensitive to Mach number for low speed flows. A more likely explanation is the nature of the modelling of the blade boundary layers by the Dawes code. As stated in Chapter 7, the Dawes code assumes turbulent flow throughout the domain. In other words, transition is not modelled. Rodger found that the suction side boundary layer at midspan remained laminar from the leading edge to about $x/C=0.5$. In Section 6.2.1.1, it was shown that the loss production in turbulent boundary layers is substantially higher than in laminar boundary layers.

8.1.1.2 Exit Flow Deviation

The exit flow deviation is the difference between the exit flow angle and trailing-edge metal angle measured in the trailing-edge plane ($\alpha_e - \beta_w$). The deviation is given in terms of a pitchwise mass-averaged value which is calculated as follows:

$$\delta = \beta_u - \frac{1}{\dot{m}} \int_0^{\dot{m}} \alpha_u d\dot{m} .$$

The results for the mass-averaged deviation at midspan are presented in Figure 8.2; computational predictions are compared to experimental measurements. The predictions follow the same general trend as the experimental results, namely that the exit flow deviation increases linearly with increasing flow convergence. The increasing trend can possibly be explained by the differences in trailing-edge blade loading for different values of AVDR. The trailing-edge flow angle is influenced by pressure differences between pressure and suction surfaces. The flow leaves the trailing edge at an angle dictated by the requirement to establish pressure equilibrium downstream. It will be shown that an increase in AVDR seems to increase the pressure differential across the trailing edge, thus influencing the exit flow angle.

The computational results underpredict the deviation by as much as $1\frac{1}{2}^\circ$. This underprediction cannot be explained by the differences in Mach number. Kiock *et al.* (1986) and Graham and Kost (1979) have shown that the exit flow underturning is less severe as the exit Mach number is increased (for subsonic flow), which is the opposite trend to that observed. As mentioned in Section 6.1.6, the H-grid has difficulty dealing with the trailing-edge region. Figure 8.3 shows a close-up view of the trailing-edge grids adjacent to the blade pressure and suction surfaces. It can be seen that the pressure and suction side grids merge at the maximum axial position of the blade. The skew of the grid around the base region can possibly have an effect on the results in that region.

8.1.1.3 Base Pressure

As mentioned in Section 6.2.1.1, the base pressure is directly related to the trailing-edge losses caused by the finite thickness of the blade trailing edge. Lower values of base pressure are associated with higher trailing-edge losses. The base pressure is presented in terms of a non-dimensional coefficient, defined as follows:

$$C_{pb} = \frac{P_b - P_2}{\frac{1}{2} \rho V_2^2}$$

where P_b is the pressure in the base region.

Due to the formulation of the Dawes code, slight pressure oscillations are present in the solution and are most apparent near the leading and trailing edges. Figure 8.3 shows that there are about eight cells which wrap around the trailing edge. It was found that, regardless of the cell chosen from which to extract the base pressure, the trend with AVDR (ie. the slope) remained unchanged, although the curve was shifted to higher or lower values. Therefore, it was decided that the base pressure would be calculated as the average pressure between cells with J indices of 77 and 78 as shown in Figure 8.3, since these two cells best approximate the location of the blade's base.

Figure 8.4 shows that the computational results give a reasonably good prediction of the experimentally found values. The base pressure is seen to rise roughly linearly with increasing AVDR, thus reducing the component of trailing-edge loss. As mentioned above, the absolute value of the predicted base pressure coefficient is somewhat uncertain due to the oscillations in the pressure field in the trailing-edge region and the relatively close agreement in results may be somewhat fortuitous.

8.1.1.4 Blade Static Pressure Distribution

The blade static pressure distributions at midspan are presented in terms of a nondimensional coefficient defined as follows:

$$C_p = \frac{P_x - P_1}{\frac{1}{2} \rho V_1^2}$$

where the subscripts x and 1 refer to the x/C_x location and inlet plane respectively.

Midspan static pressure distributions from predictions and measurements are presented in Figure

Figure 8.3: Trailing-edge grid configuration.

Figure 8.4: The effect of AVDR on the base pressure coefficient (C_{pb}) at design incidence for computational and experimental, subsonic results.

8.5. The agreement is generally good, especially on the pressure surface where the blade experiences a smooth acceleration (favourable pressure gradient). The largest discrepancy occurs on the aft portion of the suction surface. The computational results show a smooth acceleration to about $x/C_x=65\%$ followed by a smooth deceleration (adverse pressure gradient). In contrast, the experimental results showed the pressure minimum at about $x/C_x=45\%$. This discrepancy can possibly be due to Mach number effects. It is well known that as the Mach number is increased, turbine blades tend to become more loaded in the aft region (eg. Mee *et al.*, 1992).

Further differences between predicted and measured results arise near the leading and trailing edges. This is likely due to the difficulty encountered in predicting the flow near the leading edge when using an H-mesh as indicated by the oscillations in that region. In the trailing-edge region, the computational predictions show an unusual pressure minimum spike. The pressure spike usually occurs at node 73 or 74 on the pressure side (see Figure 8.3). This pressure distribution is consistent with the situation of the flow following a path of high curvature, leading to high velocity (or low pressure). This situation may also be a possible cause for the underprediction in deviation.

Figure 8.6 shows the predicted influence of AVDR on the midspan blade pressure distribution. As the value of AVDR increases, the blade becomes more loaded as expected. This is evidenced by the sharp decrease in pressure on the suction surface and a milder pressure decrease on the pressure surface. The general shape of the loading curve remains essentially the same; although, for $AVDR > 1$, there should be an overall acceleration, it is in fact predicted to produce a stronger adverse pressure gradient on the rear of the suction surface which may lead to earlier separation. Rodger (1992) observed the same general trends (not shown here) but they were not as sensitive to the AVDR, and the strength of the adverse pressure gradient on the suction surface showed little variation with AVDR. The difference between predicted and measured results can possibly be explained by the Mach number effect. In the computational tests the inlet Mach number decreased by roughly 35% from the lowest AVDR case to the highest AVDR case, thus affecting the value of C_p . The experiments were essentially incompressible. The trend in blade pressure distributions explains the general increase in losses with increasing AVDR as discussed previously in Section 8.1.1.1.

Figure 8.5: Midspan static pressure distribution at design incidence for computational and experimental, subsonic results.

Figure 8.6: The effect of AVDR on the midspan pressure distribution at design incidence for computational, subsonic results.

8.1.2 Off-Design Incidence

A wide range of operating conditions can exist in the aircraft engine: for example, idle, take-off and cruise. Consequently, the inlet flow angles can vary from the design values, thus increasing the losses. These additional losses for operation away from the design point are usually termed "off-design losses". Tremblay *et al.* (1990) investigated the off-design performance of the blade used in the present study and Rodger (1992) studied the sensitivity of the blade to the AVDR at off-design incidences (ie. $+20^\circ$ and -20°). In the present study, off-design results at incidences of $+20^\circ$ and -10° were calculated. The influence of incidence on various aerodynamic parameters: namely profile loss, exit flow deviation, base pressure and blade loading, will be presented. Investigations of -20° were not performed due to problems with convergence at this extreme case.

8.1.2.1 Profile Loss

The computed losses are presented in Figure 8.7 and the experimental results in Figure 8.8 .

For negative off-design incidence, the predicted results show virtually the same magnitude and trend of profile loss with the AVDR as for the design case. The experimental results also show similar magnitudes but the trend is somewhat different. The increase in loss for flow divergence is not as strong as it was for the design incidence case.

For positive off-design incidence ($+20^\circ$), the profile losses increase sharply for both predicted results (Figure 8.7) and measured results (Figure 8.8). The predicted results show roughly a doubling of loss for an AVDR of about unity, whereas, the measured results show roughly a seven-fold increase for the same condition. Flow visualizations performed by Rodger (1992) revealed that there was a substantial region of flow separation on the suction surface from about $x/C_x=0.5$ to the trailing edge, for the $+20^\circ$ incidence case. This would explain the large increase in profile loss. As was mentioned previously, the Dawes code assumes turbulent boundary layers on the entire blade surface; thus, it is likely that the separated region was predicted further downstream than that found experimentally. The profile loss is seen to decrease with AVDR for both computational and experimental results. This trend can possibly be linked to the effect of

Figure 8.7: The effect of AVDR on midspan profile loss (Y'_{ms}) at design and off-design incidences for computational, subsonic results.

Figure 8.8: The effect of AVDR on midspan profile loss (Y'_{ms}) at design and off-design incidences for experimental, subsonic results.

flow acceleration ($AVDR > 1$) on the position of flow separation. An increase in AVDR would tend to move the separation point further downstream and thus reduce the profile loss. However, as mentioned in Section 8.1.2.4, the trend in blade loading with AVDR (Figure 8.6) seems to contradict these implications.

8.1.2.2 Exit Flow Deviation

The results showing the effects of AVDR on the exit flow deviation are presented in Figure 8.9 for the computations and Figure 8.10 for the experiments.

For negative off-design incidence, the computational predictions at -10° incidence showed roughly the same magnitude and trend as for design incidence, whereas, the experimental results showed a slight increase in deviation (about 1°) for the -20° incidence case, but a similar trend with AVDR. Tremblay *et al.* (1990) explain the increase in deviation as being due to the increasingly large pressure difference which occurs over the rearward part of the blade at negative incidences. The computational and experimental results cannot be directly compared since they are presented for different values of incidence.

For positive off-design incidences ($+20^\circ$), both the computational and experimental results show a substantial increase in the exit flow deviation. The increase is more severe for the experimental results. The general overturning of the flow at positive off-design incidences is likely explained by the suction side separation near the trailing edge.

8.1.2.3 Base Pressure

The results showing the effects of AVDR on the base pressure are presented in Figure 8.11 for the computations and in Figure 8.12 for the experiments.

For negative off-design incidence, the computations predict similar magnitudes and trends as for the design incidence case. The experimental results also show similar magnitudes and trends.

Figure 8.9: The effect of AVDR on exit flow deviation (δ_{ms}) at design and off-design incidences for computational, subsonic results.

Figure 8.10: The effect of AVDR on exit flow deviation (δ_{ms}) at design and off-design incidences for experimental, subsonic results.

Both the computations and experiments reveal that for the negative incidence case, the base pressure increases with AVDR; consequently the trailing-edge losses are higher at lower values of AVDR.

For positive off-design incidence (+20°), the computational results predict the same trend with AVDR but lower levels of base pressure than for the attached flow cases. This result implies that the trailing-edge loss component is a more dominant factor for the +20° case. This increase in trailing-edge loss is consistent with the observation of flow separation on the suction side, as mentioned previously. Tremblay (1989) found that the trailing-edge loss accounted for about 14% of the total loss at high positive incidence compared to only 5% of the total loss at design incidence. This increase in trailing-edge loss at high positive incidences may help to explain the profile loss results presented in Figures 8.7 and 8.8. The experimental results for base pressure at +20° incidence are inconclusive since only two data points were recorded. Rodger (1992) suggested that inaccurate measurements could have been made due to the chaotic flow in the separated trailing-edge region for this case.

8.1.2.4 Blade Static Pressure Distribution

Figure 8.13 shows the predictions and Figure 8.14 shows the measurements for the variation in blade static pressure at midspan for design and off-design incidences.

For the predicted case of -10° incidence, the pressure distribution is similar to that of the design incidence case with a few exceptions. The -10° incidence shows that the distribution is shifted to lower values of pressure particularly on the suction side. Furthermore, larger pressure gradients are experienced by the -10° incidence case on the aft portion of the suction surface. The experimental results for -20° incidence show the same general trend.

For the case of +20° incidence, the pressure distribution is significantly different from that for the attached flow cases. The suction surface experiences a high overspeed region near the leading edge, followed by a steep adverse pressure gradient. This adverse pressure gradient would explain the separated region on the suction side, previously discussed. As mentioned earlier, Denton and

Figure 8.11: The effect of AVDR on the base pressure coefficient (C_{pb}) at design and off-design incidences for computational, subsonic results.

Figure 8.12: The effect of AVDR on the base pressure coefficient (C_{pb}) at design and off-design incidences for experimental, subsonic results.

Figure 8.13: Midspan static pressure distribution at design and off-design incidences for computational, subsonic results.

Figure 8.14: Midspan static pressure distribution at design and off-design incidences for experimental, subsonic results.

Figure 8.15: Midspan static pressure distribution at $+20^\circ$ incidence for computational and experimental, subsonic results.

Figure 8.16: The effect of AVDR on the midspan pressure distribution at -10° incidence for computational, subsonic results.

Figure 8.17: The effect of AVDR on the midspan pressure distribution at $+20^\circ$ incidence for computational, subsonic results.

Cumpsty (1987) have shown that loss production in a boundary layer is roughly proportional to the cube of the velocity at the edge of the boundary layer. Consequently, the overspeed experienced for flows at high positive incidences can further explain why the profile losses greatly increase at high positive incidences, as previously shown.

Figure 8.15 shows the comparison between the predicted and measured static pressure distribution at midspan for the $+20^\circ$ incidence case. In general, the results are in close agreement, especially on the pressure surface. The largest discrepancy occurs on the aft portion of the suction surface; this may be due to Mach number effects (as mentioned in Section 8.1.1.4), or it may be due to the fact that the computational results cannot predict a laminar boundary layer on the suction side, as previously mentioned. Further discrepancies arise near the leading edge and near the trailing edge (low pressure spike) for the same reasons as mentioned previously in Section 8.1.1.4.

Figures 8.16 and 8.17 show the predicted variation of midspan blade static pressure with AVDR. The trends with AVDR for the off-design cases are similar to that for the design incidence case (Figure 8.6). With increasing flow convergence, the pressures are shifted to lower values, especially on the suction surface. Similar trends were measured by Rodger (1992).

8.2 The Effects of AVDR for Transonic Conditions at Design and Off-Design Incidences

In recent years, turbine designers have opted to decrease the number of turbine stages to reduce the size and weight of the engine. The resulting stages are subject to increased aerodynamic loading and transonic operating conditions which lead to increased complexity and higher losses, mainly in the form of shock losses. There is a great need for experimental testing of transonic turbine cascades to obtain an understanding of the physical flow processes present. An understanding of the effects of the AVDR on turbine performance parameters is needed for the experimenter to be aware of the effects of lack of flow two-dimensionality on measurements.

This section presents the predicted effects of the AVDR on turbine blades operating at low and high-transonic conditions: namely at exit isentropic Mach numbers of $M_{2s}=0.98$ and $M_{2s}=1.15$. Calculations have been performed at design incidence, $+10^\circ$ incidence, and -10° incidence. There are essentially no measurements available and it is therefore not possible to estimate the uncertainty of the predictions.

In Section 8.3 comparisons of the predicted results for the three Mach number regimes will be investigated.

8.2.1 Profile Loss

Figures 8.18 and 8.19 show the predicted effects of the AVDR on the profile loss for the low transonic and high-transonic cases respectively.

For the low-transonic case at design incidence, the profile loss is seen to decrease with AVDR. The trend in profile loss becomes more sensitive to the AVDR at lower values of AVDR. At -10° incidence, the profile loss decreases linearly with AVDR and the magnitude is roughly the same as for the design incidence case, as was the result for subsonic conditions. At $+10^\circ$ incidence, the trend of profile loss with AVDR is very similar to that of the design incidence case but the magnitudes of loss are slightly greater. Recall that the losses also rose for the positive incidence case at subsonic conditions but no comparison can be made due to the difference in incidence angle for the two cases.

For the high-transonic case, the profile loss is seen to decrease linearly with the AVDR at roughly the same slope and magnitude for all three incidences. The losses are seen to be significantly more sensitive to AVDR than for the low-transonic cases. The results will be further explained in the following sections on base pressure and blade pressure distribution. The differences between the low-transonic and high-transonic cases will be discussed in Section 8.3 .

Figure 8.18: The effect of AVDR on midspan profile loss (Y'_{ms}) at design and off-design incidences for $M_2=0.98$.

Figure 8.19: The effect of AVDR on midspan profile loss (Y'_{ms}) at design and off-design incidences for $M_2=1.15$.

8.2.2 Exit Flow Deviation

Figures 8.20 and 8.21 show the effects of the AVDR on the deviation angle for the low-transonic and high-transonic cases respectively.

For the three low-transonic cases the exit flow deviation is seen to increase with the AVDR, as was found for the subsonic test cases. Both for the design incidence case and the -10° incidence case, the change in deviation angle is more sensitive to the AVDR at lower values of AVDR. For the $+10^\circ$ incidence case, the trend is linear but the magnitudes are similar to the other two cases.

For the three high-transonic cases the exit flow deviation decreases linearly with the AVDR at approximately the same slope and magnitude. The differences in trend between the low-transonic and high-transonic cases will be discussed in Section 8.3 .

8.2.3 Base Pressure

Figures 8.22 and 8.23 show the effects of the AVDR on the base pressure for the low-transonic and high-transonic cases respectively.

All three low-transonic cases show a linear increase in the base pressure as a function of the AVDR at about the same slope. The -10° incidence case predicts base pressures slightly higher than those predicted for the design incidence case, whereas, the $+10^\circ$ incidence case predicts base pressures slightly lower than those predicted for the design incidence case. As previously mentioned (in Section 6.2.1.1), the trailing-edge loss component becomes more important for transonic flow conditions contributing about 1/3 to the total losses. Furthermore, the base pressure value gives a good indication to the relative level of trailing-edge loss whereby lower values of base pressure correspond to higher levels of loss. Therefore, it is possible that the larger trailing-edge component of loss for the $+10^\circ$ incidence case can explain the higher profile loss predicted at this incidence. Also, for all three incidence cases, the profile losses are higher for conditions experiencing flow divergence ($AVDR < 1$), as seen in Figure 8.18 .

For the three high-transonic cases the base pressure coefficient increases with AVDR at roughly

Figure 8.20: The effect of AVDR on exit flow deviation (δ_{ms}) at design and off-design incidences for $M_2=0.98$.

Figure 8.21: The effect of AVDR on exit flow deviation (δ_{ms}) at design and off-design incidences for $M_2=1.15$.

Figure 8.22: The effect of AVDR on the base pressure coefficient (C_{pb}) at design and off-design incidences for $M_2=0.98$.

Figure 8.23: The effect of AVDR on the base pressure coefficient (C_{pb}) at design and off-design incidences for $M_2=1.15$.

the same slope. Furthermore, the magnitudes of base pressure coefficient are approximately equal for the three incidence cases; hence the trailing-edge losses are also likely to be similar in magnitude which may possibly explain the similar levels in total pressure loss for the three incidence cases shown in Figure 8.19 .

8.2.4 Blade Static Pressure Distribution

Figures 8.24 and 8.25 show the variation of the blade static pressure distribution with the AVDR at design incidence for the low-transonic and high-transonic cases respectively. The results show that as the AVDR is increased the blade loading is increased; the pressure surface values decrease slightly and the suction surface values decrease substantially. Furthermore, as the AVDR is increased, the suction surface encounters a stronger adverse pressure gradient which may eventually lead to separation and higher losses at higher values of AVDR. As mentioned earlier, it would be expected that the profile losses would be greater for the flow convergence cases since the higher velocities through the blade row would lead to higher entropy or loss production in the boundary layers. It appears that the trailing-edge component of loss becomes a more dominant factor for the transonic case. Similar trends in blade pressure distribution with AVDR were predicted for the off-design cases (but are not shown here).

Figures 8.26 and 8.27 show the change in blade static pressure distribution for design and off-design incidences at AVDR values near unity. The same general trends are predicted as were found at subsonic conditions. The blade loading is highest for the -10° incidence case and is lowest for the $+10^\circ$ incidence case. Furthermore, the blades become more forward loaded as the incidence is increased. A slight overspeed region on the suction side near the leading edge is observed for the $+10^\circ$ incidence case. It is not as dominant a feature as was shown for the subsonic cases; but this may be due to the lower value of positive incidence tested for the transonic cases. Recall that for the subsonic case, the positive incidence test was performed at $+20^\circ$ incidence. Tremblay *et al.* (1990) have shown that the suction surface overspeed becomes a dominant feature at $+20^\circ$ incidence and is hardly noticeable at $+10^\circ$ incidence for subsonic conditions. Therefore, it is possible that the losses would increase substantially if the incidence was increased higher than $+10^\circ$.

Figure 8.24: The effect of AVDR on the midspan static pressure distribution at design incidence for $M_{2s}=0.98$.

Figure 8.25: The effect of AVDR on the midspan static pressure distribution at design incidence for $M_{2s}=1.15$.

Figure 8.26: Midspan static pressure distribution at design and off-design incidences for $M_2=0.98$.

Figure 8.27: Midspan static pressure distribution at design and off-design incidences for $M_2=1.15$.

8.3 Mach Number Effects

In this section, results are presented for the design incidence cases at the three values of Mach number studied ($M_{2s} = 0.6, 0.98, 1.15$). As mentioned earlier, the Mach number can have a significant affect on the performance of turbine blades, especially when they are operating at transonic conditions. The effects of AVDR on profile loss, exit flow deviation, base pressure, and blade static pressure distribution are presented for the design incidence cases for all three Mach numbers examined.

8.3.1 Profile Loss

Figure 8.28 shows the variation of profile loss with AVDR at design incidence for the three different Mach number conditions.

The first feature to notice is the relative levels of loss at the different Mach number conditions. In general, the losses for the transonic cases are slightly higher than for the subsonic cases, except at high values of AVDR. The high transonic cases show a much sharper increase than the levels observed at low transonic conditions. These results generally agree with those of Mee *et al.* (1992), presented in Section 6.2.1.

The trends in profile loss with AVDR are different for the three Mach number conditions. For the subsonic case, the profile loss linearly increases with the AVDR. For the low transonic case, the loss decreases slightly with the AVDR. For the high-transonic case, the loss decreases linearly with the AVDR, and at a steeper slope than for the low transonic case. The trends will be explained further in the following sections. The results imply that flow convergence or divergence becomes a less important effect at Mach numbers near unity but becomes very important for higher levels of Mach number.

Figure 8.28: The effect of AVDR on midspan profile loss (Y'_{me}) at design incidence for subsonic and transonic conditions.

Figure 8.29: The effect of AVDR on the exit flow deviation (δ_{me}) at design incidence for subsonic and transonic conditions.

8.3.2 Exit Flow Deviation

Figure 8.29 shows the effects of AVDR on the exit flow deviation for the different Mach number cases at design incidence.

In general, the magnitude of deviation is seen to decrease from the subsonic case to the low-transonic case and increases substantially as the Mach number is increased past unity except at high AVDR. This trend is expected since Ainley and Mathieson (1951) have shown that the deviation approaches zero as the Mach number tends towards unity. Graham and Kost (1979) and Kiock *et al.* (1986) presented similar results; the magnitude of the exit flow deviation was seen to be a minimum at Mach numbers near unity and then increased rapidly for higher Mach numbers. The generally high levels of deviation might possibly be explained by the configuration of the computational grid near the trailing edge, as mentioned earlier.

The trends of exit flow angle with AVDR are different for the three Mach number cases. For the subsonic case, the deviation increases linearly with AVDR. For the low-transonic case, the deviation increases linearly with AVDR but the dependence is much weaker than for the subsonic case. For the high-transonic case a strong linear dependence of deviation with AVDR is observed. Unlike for the other two cases, the deviation decreases with AVDR for the high-transonic case.

8.3.3 Base Pressure

Figure 8.30 shows the effects of AVDR on the base pressure for the different Mach number cases at design incidence.

It can be seen that the value of the base pressure coefficient increases from the subsonic cases to the low-transonic cases and then decreases to a lower level for the high-transonic cases. This trend was also observed by Kiock *et al.* (1986); they found that the base pressure coefficient increased slightly with Mach number for exit Mach numbers less than about 0.9, at which point the pressure increased due to a normal trailing-edge shock. For further increases in exit Mach number, the normal shock developed into an oblique shock which resulted in a rapid decrease in base pressure. Generally speaking, a lower base pressure indicates a higher trailing-edge loss contribution at a given Mach number. However, the trailing-edge contribution to loss is more

Figure 8.30: The effect of AVDR on the base pressure coefficient (C_{pb}) at design incidence for subsonic and transonic conditions.

Figure 8.31: Midspan static pressure distributions at design incidence for subsonic and transonic conditions (AVDR=1).

important as Mach number increases (eg. Xu and Denton, 1988 and Denton and Xu, 1990).

The base pressure coefficient is seen to increase linearly with AVDR for all three Mach number cases. The trend is stronger for the high-transonic case than for the other two cases. The results imply that for all Mach numbers investigated, the base pressure is lower and hence the trailing-edge loss is higher for lower values of AVDR. As mentioned earlier, the trailing-edge loss component becomes increasingly important at higher Mach numbers. Therefore, the trend in base pressure coefficient for the transonic cases may possibly explain the trends in profile loss presented in Figure 8.28 . As mentioned earlier, the profile losses due to entropy production in the boundary layers increase with flow acceleration through the blade passage ($AVDR > 1$). The trailing-edge loss component has the opposite effect with AVDR and begins to dominate the losses at higher Mach numbers. Thus the predicted profile losses are seen to increase with AVDR for low speed flow but tend to decrease for high speed flow as seen in Figure 8.28 .

8.3.4 Blade Static Pressure Distribution

Figure 8.31 presents the predicted effect of exit Mach number on the blade static pressure distribution for design incidence cases. It can be seen that the blade becomes more loaded with increasing Mach number. The pressure distributions on the pressure surface are very similar for all three cases. The pressure is roughly constant up to about $x/C_x \approx 0.7$ at which point it rapidly decreases. The suction surface distribution especially on the aft portion shows major differences for the different cases, namely, that the blade becomes more aft loaded as the Mach number is increased. This is a well known effect for turbine blades operating at transonic conditions (eg. Mee *et al.*, 1992).

Conclusions and Recommendations

9

9.1 Conclusions

A computational investigation of the effects of AVDR on the midspan performance parameters of a linear turbine cascade was performed using a three-dimensional Navier-Stokes solver, the Dawes code. A detailed description of the solver has been presented, and its usefulness as an analysis tool for turbomachinery research has been investigated. Specifically, the code has been validated against experimental results, performed by Rodger (1992), for the effects of axial velocity-density ratio (AVDR) on turbine blade performance at subsonic conditions. Furthermore, a study of the effects of AVDR at transonic conditions has been conducted, as an extension of Rodger's work. In general, it has been found that the Dawes code is a useful tool for turbomachinery analysis. Although quantitative predictions were not very accurate, the basic trends were predicted reasonably well. These trends can be used to gain physical insights into the important flow processes influencing turbine blade performance at both subsonic and transonic conditions.

Specific conclusions regarding the Dawes code are as follows:

- (i) The structure of the Dawes code (ie. implicit solver, **H**-grid, Baldwin-Lomax turbulence model) makes it a simple, economic tool which can easily be used for a wide range of turbomachinery problems, providing reasonably accurate qualitative and sometimes quantitative predictions.
- (ii) The configuration and density of the structured **H**-grid must be carefully chosen to ensure reasonably grid-independent solutions. Consideration of the relative grid spacing is important to avoid high residuals and spurious oscillations in the solution.
- (iii) The user of the code must be certain not to terminate the solution until reasonable convergence (to within machine accuracy) is achieved. RMS residuals are generally not a reliable indication of convergence. Flow quantities of interest (such as loss) should be monitored along with residuals and mass flow rates. It is always good practice to be generous with the number of iterations.
- (iv) The accuracy of solutions will decline for very low (incompressible) Mach number flows, for flows where the thin-layer approximation breaks down, and for flows where laminar boundary layers are a dominant feature.

Specific conclusions regarding the effects of AVDR on turbine blade performance are as follows:

- (i) Increasing values of AVDR result in an acceleration of flow through the blade passage, resulting in a shift of the pressure distributions to lower values and an increase in losses due to entropy production in the blade boundary layers.
- (ii) Increasing values of AVDR result in an increase in base pressure values and a corresponding decrease in trailing-edge losses due to the finite thickness of the trailing edge.
- (iii) As Mach number is increased to transonic values, the importance of the boundary-layer losses decrease relative to trailing-edge losses. Trailing-edge losses become the dominant component for turbine blades operating at transonic conditions.
- (iv) The exit flow deviation can be significantly influenced by AVDR. For subsonic conditions ($M_{2a}=0.6$) it tends to increase with AVDR whereas for high transonic conditions ($M_{2a}=1.15$) it tends to decrease with AVDR.

- (v) For subsonic operating conditions ($M_{2s}=0.6$), negative incidences (-10°) have little effect on performance, whereas positive incidences ($+20^\circ$) have significant effects: increasing the profile losses and decreasing the deflection due to suction surface boundary-layer separations.
- (vi) For low-transonic conditions ($M_{2s}=0.98$), negative incidence (-10°) has a minor effect on turbine blade performance, and positive incidence ($+10^\circ$) tends to increase the profile losses marginally, while having little effect on exit flow deviation.
- (vii) For high-transonic conditions ($M_{2s}=1.15$), off-design incidences (-10° and $+10^\circ$) were found to have virtually no effect on the performance of the turbine blade tested.

Hence, the results show that AVDR effects are important to consider, especially at high-transonic Mach numbers. Profile losses and deviation angles, reported in the literature, should be viewed with caution since most studies do not report the value of AVDR present in their experiments.

9.2 Recommendations

Several recommendations are presented for the operation and modification of the Dawes code. They are as follows:

- (i) The major shortcoming of the Dawes code is that it assumes fully turbulent flow throughout. Most of the research presently being conducted at Carleton University regarding turbomachinery aerodynamics is related to axial turbines with relatively low chord Reynolds numbers. Turbine blades are known to experience significant regions of laminar flow on the suction surface. Laminar boundary layers significantly influence the performance of turbine blades by reducing the skin friction and by being more susceptible to separation. The code will become significantly more useful if a simple transition model can be incorporated into the code although this is not a trivial task.

- (ii) The Baldwin-Lomax model has problems in the blade/endwall corner regions as evidenced by the unrealistic negative losses predicted at that location. This problem is likely due to the lack of a blending function in the Baldwin-Lomax model. It would be straightforward to add such a blending function, as discussed in Section 7.1.5. Note that this deficiency was not a concern in the present work since midspan results were of interest.
- (iii) The inviscid endwall option is not implemented correctly in the current version of the code. Correcting this problem would be useful to potential users; it would allow modelling "half cascades" by using an inviscid wall as the plane of symmetry, thus reducing the number of cells necessary to model the domain, resulting in a reduction in computation time.
- (iv) The code is a useful tool for analyzing turbine cascades operating at transonic conditions. The code must be validated for some experimental transonic cases. This can be done by performing transonic AVDR experiments or more easily by running the code for cases which are thoroughly documented in the open literature.

Recommendations for further study on the effects of AVDR on turbine blade performance are as follows:

- (i) The computational results for transonic operating conditions presented in this work must be validated by experiment.
- (ii) Investigations should be conducted for a wider range of Mach numbers around and above unity since it has been seen that turbine blade performance is very sensitive to Mach numbers in the transonic regime.
- (iii) Transonic investigations should be conducted at higher values of positive incidence where results are likely to change rapidly.

Wind Tunnel Contraction Coordinates

Figure 1A.1: Contraction Geometry

Table 1A.1: Contraction Coordinates

x (in)	r (in)	x (in)	r (in)
0.00	8.000	7.75	5.575
0.25	8.000	8.00	5.470
0.50	7.999	8.25	5.310
0.75	7.998	8.50	5.162
1.00	7.995	8.75	5.023
1.25	7.990	9.00	4.894
1.50	7.982	9.25	4.775
1.75	7.972	9.50	4.664
2.00	7.958	9.75	4.563
2.25	7.941	10.00	4.470
2.50	7.919	10.25	4.385
2.75	7.892	10.50	4.307
3.00	7.859	10.75	4.237
3.25	7.821	11.00	4.174
3.50	7.777	11.25	4.118
3.75	7.725	11.50	4.067
4.00	7.667	11.75	4.023
4.25	7.600	12.00	3.984
4.50	7.525	12.25	3.950
4.75	7.442	12.50	3.921
5.00	7.349	12.75	3.897
5.25	7.246	13.00	3.876
5.50	7.191	13.25	3.859
5.75	7.133	13.50	3.846
6.00	7.010	13.75	3.835
6.25	6.875	14.00	3.827
6.50	6.728	14.25	3.821
6.75	6.569	14.50	3.817
7.00	6.398	14.75	3.815
7.25	6.213	15.00	3.814
7.50	6.015	15.25	3.813
	5.802	15.50	3.813

Appendix

1B

Physical Properties of MB1000 and Renshape

	<u>MB1000</u>	<u>Renshape</u>
Density	39.06	35.89
Heat Distortion (66 psi)	158°F	164°
Tensile Properties (psi)		
tensile strength	2663.3	2250
tensile modulus	45897	30810
Flexural Properties (psi)		
flexural strength	4250	3790
flexural modulus	128081	77832
Compressive Properties (parallel)		
10% deflection (psi)	3289	3118
compression modulus (psi)	127200	86693
Izod Impact		
average ft-lbs/inch	1.60	1.11
Coefficient of Linear Thermal Expansion		
inch/inch/°F	42.76×10^{-6}	58.8×10^{-6}
Dimensional Stability % Volume Change		
158°F 100% RH		
1 Day	2.0	2.8
2 Days	1.1	2.6
7 Days	1.4	2.6
14 Days	3.0	3.1
200°F		
1 Day	2.3	2.7
2 Days	0.7	1.8
7 Days	0.5	-5.9
14 Days	-1.2	-1.4
-20°F		
1 Day	-0.8	-0.2
2 Days	-1.5	-1.2
7 Days	-2.2	-1.8

Physical Properties of BC-5003

Physical Properties

Specific Gravity	1.14
Hardness, (Shore D) @ 75°F	86
Tensile Strength (Ultimate)	11,200 psi
Flexural Strength (Ultimate)	19,000 psi
Flexural Modulus	4.8×10^5 psi
Compressive Strength (Ultimate)	16,800 psi
Heat Distortion Temperature (264 psi)	172°F

Working Properties

Mix Ratio (Resin/Hardener)	100 to 20 by wt. 100 to 22 by vol.
Viscosity (Mixed) Brookfield @ 75°F	3,500 cps
Pot Life (1/2 lb. mass) 75°F	14-15 Minutes
Cure Schedule @ 75°F	6 to 8 Hours

Description of the Navier-Stokes Solver, BTOB3D

A description of the program BTOB3D and each of its subroutines is presented. A description of the calculations made in each subroutine is presented chronologically to allow interested readers to follow and identify operations in the code. Equations are identified and cross referenced to the equations presented in Chapter 7.

Main Program

- Summary:** The main program initiates the computations and controls the interaction between fine and coarse mesh solvers.
- Calls:** *Startup, Solver, Nextmesh, Restrict, Collect, Solverg, Prolong, Bcsio, Output*
- Description:**
- initializes input and output files
 - calls *Startup* to initialize geometry and flow variables
 - initiates main iteration loop: first calls *Solver* to solve the governing equations on the fine grid
 - multigrid loop calls *Nextmesh, Restrict, Collect, Solverg, Prolong & Bcsio* and then returns to main loop
 - uses the equation of state to update the pressure and rothalpy for all points in the domain
 - calls *Output* to dump results to output files

Startup

- Summary:** This subroutine reads the input file produced by the pre-processor and calculates the grid geometry. It also initializes the initial flow field.

Called by: *Main Program*

Calls: *Areah, Intp*

Description:

- reads in geometric data and control variables from the input file
- generates blade-blade and meridional meshes (ie. calculates the coordinates of the vertices of each hexahedral mesh) and sends data to first output file
- modifies mesh for twisted blades or tip clearance
- calls *Areah* to work out projected areas of cell faces
- initializes flow field
- allows restart from graphics dump (this feature is not implemented)
- sets value of timestep for each element based on the CFL number

Areah

Summary: This subroutine is used by the fine grid solver and computes the projected areas of the cell faces and volumes of the cells.

Called by: *Startup, Solver*

Description:

- uses the vector product of the diagonals of a cell face to calculate the three components of area: axial, radial, and tangential (see Equation 7.2)
- quasi-orthogonal face - area of axial projection = AQX (main component), area of radial projection = AQR: AQR = 0 if the faces are orthogonal (ie. x is not a function of k), and AQT = 0 always because the quasi-orthogonal faces are surfaces of revolution
- bladewise face - area of the tangential projection = ABT (main component), area of the axial projection = ABX (occurs due to variable thickness of blades and blade turning), & area of the streamwise projection = ABR (only occurs for twisted blades or blades with tip clearance);
- streamwise face - area of radial projection = ASR (main component), area of the axial projection = ASX (occurs for non-constant radius hub and casing)
- volume of each cell is calculated according to Equation 7.3

Areag

Summary: This subroutine is used by the coarse grid solver and computes the projected areas of the cell faces and volumes of the cells similar to *Areah*. The variables in this subroutine share the same common block with those of *Areah* to save memory and hence both sets (fine and coarse) are continuously recalculated.

Called by: *Solverg*

Nextmesh

Summary: Takes the geometric information of the fine grid and deletes every second line in each coordinate direction to establish the geometry for the coarse mesh.

Called By: *Main Program*

Restrict

Summary: Transforms the fine grid flow variables to the coarse grid.

Called By: *Main Program*

Description: • volume averages the flow variables and time step from eight fine cells which make up one coarse cell (see Equation 7.15)

Collect

Summary: Calculates forcing functions from fine grid to be appended to governing equations of coarse grid solver, *Solverg*.

Called By: *Main Program*

Description: • forcing functions are solved from the fine grid residuals according to Equation 7.16

Prolong

Summary: Interpolates corrections from coarse grid to fine grid.

Called By: *Main Program*

Description:

- correction terms are calculated according to Equations 7.17 and 7.18 and appended to fine grid solutions of the fine grid flow variables according to Equation 7.19

Solver

Summary: This is the main solver for the fine mesh. All inviscid terms are internally calculated and additional terms such as physical and numerical viscosity are called from other subroutines. All terms are collectively lumped into a statement of control volume conservation. Two explicit steps are performed and the final implicit step is computed in an external subroutine. This subroutine also computes RMS residuals and sends the results to an output file.

Called By: *Main Program*

Calls: *Areah, AV, NS, AF, Bcsio*

Description:

- calls *Areah* to calculate areas and volumes of cells
- calls *AV* to calculate the artificial dissipation terms which are stored as DRO, DRE, DI, DJ, and DK for each of the flow variable in vector *U* (Equation 7.9)
- calls *NS* to calculate the viscous stresses and append them to the three components of momentum: terms DI, DJ, and DK
- solves continuity equation to update density ρ in complete domain: the mass fluxes are computed through each of the 6 faces, 2 quasi-orthogonal, 2 bladewise, and 2 streamwise according to

$$\dot{m}_Q = \bar{\rho} V_x A_{Qx} + \bar{\rho} V_r A_{Qr}$$

$$\dot{m}_B = \bar{\rho} V_x A_{Bx} + \bar{\rho} V_r A_{Br} + \bar{\rho} V_z A_{Bz}$$

$$\dot{m}_S = \bar{\rho} V_x A_{Sx} + \bar{\rho} V_r A_{Sr}$$

- the first explicit step is solved when MSTEP=1 according to Equation 7.5
- the second explicit step is solved when MSTEP=2 according to Equation 7.6
- solves energy equation to update ρE at all points in the computational domain: the energy fluxes are given by the mass flux through each cell face multiplied by the rothalpy associated with that face; for example, for the quasi-orthogonal face

$$\dot{m}_Q \bar{I}$$

- two explicit steps are performed for the energy equation
- solves x-momentum equation to update ρV_x for entire computational domain: the momentum fluxes are added to the forces acting on the surface of the control volume - for example through the quasi-orthogonal face

$$\bar{p} A_{Qx} + \dot{m}_Q V_x ;$$

terms from each of the six faces are added to the viscous stress and artificial dissipation terms, DI, calculated in AV and NS respectively

- two explicit steps are performed for the x-momentum equation
- similar procedures are performed for the r-momentum and θ -momentum equations, however, body force terms (see Equation 7.1) are added to account for the rotating frame of reference
- AF is called to solve the implicit equation (Equation 7.7) for mass, energy, and three momentum equations
- advanced time step flow variables U are updated according to Equation

7.8

- **U** is decomposed to solve for the primitive variables and the pressure and rothalpy are also updated for all grid points
- *Bcsio* is called to apply inflow and outflow boundaries
- the numerical entropy source is calculated - this is only done when the code is run in diagnostic mode, ie. KENT=1
- laminar viscosity is updated
- an iteration history is performed to solve for the residuals. these results are sent to the 2nd, 3rd, and 4th output files

Bcsio

Summary: This subroutine applies boundary conditions at inflow and outflow boundaries.

Called By: *Main program, Solver*

Description:

- applies the outflow boundary condition by linear extrapolation of flow variables according to Equation 7.11
- resets pressure at downstream boundary using radial equilibrium (Equation 7.10) or maintains constant values if IPOUT=1
- applies inflow boundary conditions for all flow variables assuming inlet meridional pitch angle is fixed and specified. ie. INVX=1. INVR=1; otherwise axial and radial velocity obtained from $dV/dm = 0$ and meridional and radial velocity respectively

Solverg

Summary: This is the main solver for the coarse mesh. It is directly analogous to *Solver*, however, viscous stresses are not calculated and the fine grid artificial dissipation model is replaced by a simple scaled second order model. Two explicit steps are performed and the implicit step is calculated in an external subroutine.

Called By: *Main Program*

Calls: *Areag, AF*

Description:

- artificial dissipation is solved by a simple scaled second-order model; for

example, for the x-momentum term is

$$DJ = SCALE \{ (\rho V_x)_{i-1,j,k} + (\rho V_x)_{i+1,j,k} + (\rho V_x)_{i,j-1,k} + (\rho V_x)_{i,j+1,k} + (\rho V_x)_{i,j,k-1} + (\rho V_x)_{i,j,k+1} - 6(\rho V_x)_{i,j,k} \}$$

where $SCALE = AV2H * FT * Vol / Step$ (AV2H is usually set to zero)

- mass, energy, and three momentum equations are solved in a similar way as in *Solver* however forcing functions are used to transfer residuals from the fine grid to the governing equations in the coarse grid (see Equations 7.15-7.18)
- *AF* is called to solve the implicit equation (Equation 7.7) and the flow variables *U* are updated according to Equation 7.8
- *U* is decomposed to solve for the primitive variables on the coarse mesh and pressure and rothalpy are also calculated
- the iteration history is determined and output only if the coarse mesh is run uncoupled from the fine mesh solver in the early transient part of the solution (this is usually not done)

AV

Summary: This subroutine computes the artificial dissipation (viscosity) terms for the equations of mass, momentum and energy.

Called By: *Solver*

Description:

- artificial dissipation is calculated for i, j, and k directions for all the conservation equations
- fourth-difference dissipation terms are used to inhibit odd-even point decoupling of solutions, whereas second-difference dissipation terms are used for oscillation-free shock capturing (see Equation 7.9)
- artificial dissipation is set to zero in directions normal to solid boundaries at cells adjacent to walls to avoid masking the physical viscosity

NS

Summary: This subroutine solves for the viscous stresses and also calculates the eddy viscosity using the Baldwin-Lomax model. The Log-Law is used to calculate the wall shear stress in nodes adjacent to solid boundaries. The inviscid endwall option can be implemented.

Called By: *Solver*

Description:

- viscous stresses (ie. diffusion terms) are computed by defining a local curvilinear coordinate system and the chain rule; the diffusion terms are added to the artificial viscosity terms for each cell in the computational domain
- numerical entropy (PHI) is calculated
- Baldwin-Lomax mixing length model is used to calculate the eddy viscosity (see Equations 7.20-7.22)
- the wall shear stress is calculated using the Log-Law (see Equation 7.13); subroutine WF, contained in NS, calculates the skin-friction coefficient based on the Log-Law correlation

AF

Summary: This subroutine solves the implicit equation (Equation 7.7). The matrix inversion is done in other subroutines.

Called By: *Solver, Solverg*

Calls: *Vtri, Vtrip*

Description:

- solves the implicit equation (Equation 7.7); the equation is approximately factored and split into j, k, and i components for easy inversion
- sets up terms in the tri-diagonal matrix which are passed to *Vtri* for the j, and k sweeps or to *Vtrip* for the i sweep

Vtrip, Vtri

Summary: Inverts the tri-diagonal matrix using the Thomas algorithm. *Vtrip* is used for the i sweep whereas *Vtri* is used for the j and k sweeps. There is a slight difference

for i depending on whether the sweep is taking place inside or outside the blade passage.

Called By: *AF*

Output

Summary: This subroutine calculates the entropy and the relative Mach number. Results are sent to the output files.

Called By: *Main Program*

Description:

- calculates entropy (which is not used anywhere nor is it output) and relative Mach number
- dumps V_x , V_y , V_z , M , P , Density, Φ , and μ to the output file
- appends Y_{max} and F_{max} to the output file: these variables are used in the turbulence model

Intp

Summary: This subroutine interpolates a given set of data, y_n as a function of x_n to find the value of y for a given value of x .

Called By: *Startup*

Description of the Output Files from BTOB3D

Several files are output from BTOB3D including a file containing the grid geometry, two files containing information regarding convergence history, and results files containing the flow variables. The user can specify a maximum of five results output files at different time step values. The results output files are named in the code internally as output1 to output5. The names of the output files containing geometrical data and convergence history information are specified by the user.

The user should be warned that any variable names used in BTOB3D are likely to have local definitions only. For example, the variable GR represents entropy in subroutine *Output* and represents the $r\theta$ coordinate of grid nodes in subroutine *startup*.

Results Files ("Output1" to "Output5")

The results output files (unit 14) are produced in subroutine *Output* and contain eight columns of data in the following order: axial velocity (VX), tangential velocity (VT), radial velocity (VR), relative Mach number (GT), static pressure (P), static density (RO), numerical entropy (PHI), and effective viscosity (VMU). The variables are printed for every cell in the computational domain; ie. $K = 1$ to $KM-1$, $J = 1$ to $JM-1$, $I = 1$ to $IM-1$.

Length scale terms Y_{max} and F_{max} (corresponding to suction, pressure, hub and tip surfaces), calculated in the Baldwin-Lomax algorithm are appended to the output files.

A final note regarding the output of flow variables: in subroutine *Output* the thermodynamic property, entropy, is calculated for each cell in the computational domain and stored in GR. GR is not printed in any output file, however it is straightforward to replace PHI by GR or to simply add a ninth column in the output file if the entropy is of interest.

First Output File (Geometrical Data)

The first output file (unit 14) is printed from subroutine *Startup* and contains all geometrical data. The first four columns contain the following information: the axial coordinate (X), the radial coordinate (R), the meridional extent of elements (SMERID), and the local blade to blade pitch (RR). These variables are printed for every cell in the meridional plane; for $K = 1$ to KM and $J = 1$ to JM . Recall (from Section 7.1.2) that the streamwise and quasi-orthogonal surfaces are surfaces of revolution. Therefore, the above mentioned variables are not a function of tangential coordinate (I). The $r\theta$ coordinate (GR) of each cell in the computational domain is then appended to the first output file (for $K = 1$ to KM , $J = 1$ to JM , $I = 1$ to IM).

Second Output File (Convergence History and Trouble-Shooting)

The second output file (unit 16) contains various information which is typically investigated in the occurrence of computational runs which became unstable or produced erroneous results. The first part of this output file prints out the BTOB3D input file (see Appendix 2C) from subroutine *startup* as it was read in. This information is used to verify that the input file data was read correctly. The program then prints the following information (in five columns) for each quasi-orthogonal surface ($J = 2$ to $JM-2$) every 100 time steps from subroutine *solver*:

- the quasi-orthogonal surface index - J
- the mass flow rate through each quasi-orthogonal surface - $SUM2$
- the percent error of mass flow rate through each quasi-orthogonal surface - SM
ie. the ratio of $SUM2_J$ to the mass flow rate through the inlet quasi-orthogonal plane ($|SUM2_J/SUM2_{j=2} - 1| \times 100$)
- the fully mass-averaged stagnation pressure loss for each quasi-orthogonal surface - $SUM3$
- the percent stagnation pressure loss - SL
($|1 - SUM3/SUM3_{j=2}| \times 100$)

The program prints out additional information every five time steps in twelve columns as follows:

- the time step index - N
- the rms residue for the entire computational domain (see Section 7.2.3) - RMS
- the maximum local rms residue - $RMSM$ at cell index $IMAX$, $JMAX$, $KMAX$
- the maximum per cent error of mass flow rate through a quasi-orthogonal surface - $ECONT$ ($=\max(SM)$)
- lowest pressure in computational domain - $PSUCK$ at cell index $IMAP$, $JMAP$, $KMAP$
- the mass flow rate through the inlet quasi-orthogonal surface - $SUM1$ ($= SUM2_{j=2}$)

Lastly, the laminar viscosity (constant for the entire computational domain at a given time step) is printed every fifty time steps.

Third Output File (Convergence history)

The third output file (unit 18) contains information regarding convergence history and is printed from subroutine *solver*. This output file is useful for investigating the evolution of selected quantities for the determination of steady-state convergence. The information is output in four columns every five time steps as follows: time step index (N), rms residue (RMS), maximum per cent error of mass flow rate through the quasi-orthogonal surfaces (ECONT), and the lowest pressure in the computational domain (PSUCK).

Appendix

2B

Carleton Cascade Two (CC2) Blade Coordinates

Table: 2B.1: CC2 blade coordinates.

SUCTION SURFACE		PRESSURE SURFACE	
x	y	x	y
0.00000	23.89506	149.26590	87.08203
0.12096	23.00040	149.21585	87.62842
0.41083	22.14745	149.02607	88.14144
0.85503	21.36124	148.77909	88.58772
1.19496	20.88159	148.28574	88.93808
1.89776	19.93479	147.78940	89.16539
3.00096	18.56048	147.24927	89.25715
4.54836	16.80454	146.70289	89.20501
6.18544	15.13618	146.18987	89.01315
8.78808	12.80673	145.74358	88.69616
12.51060	10.02057	145.39531	88.27282
16.47921	7.60354	145.21179	87.98086
20.63133	5.51392	144.84058	87.40110
24.94195	3.77257	144.28168	86.53564
29.37353	2.36490	143.14094	84.83392
33.89686	1.29089	141.58520	82.60874
38.47442	0.52762	140.01068	80.42737
45.40853	0.00000	138.41532	78.28353
52.33847	0.17518	135.96283	75.14701
61.43728	1.47024	133.43318	72.10226
70.22327	3.91230	129.93172	68.17536
78.54212	7.36163	126.27384	64.40279
86.27496	11.65557	122.44496	60.77619
93.35714	16.52926	118.42630	57.31227
99.77406	21.77624	114.21996	53.99432
105.56326	27.17964	109.80089	50.84738

SUCTION SURFACE		PRESSURE SURFACE	
x	y	x	y
110.79148	32.60806	105.17328	47.85268
115.53587	37.95931	100.31419	45.04566
119.86318	43.21464	95.22570	42.39298
123.84638	48.35318	89.88070	39.94466
127.54179	53.38120	84.27293	37.68612
131.01406	58.30494	78.38571	35.64447
134.31116	63.14943	72.19819	33.81344
137.48521	67.93554	65.69577	32.22433
139.80005	71.51833	58.84298	30.86462
142.08988	75.09905	53.46253	30.03461
143.60600	77.49940	47.85893	29.36101
145.11169	79.91643	43.98209	28.98563
146.64449	82.34589	39.99263	28.69367
147.77898	84.19160	35.87178	28.46010
148.35877	85.11545	31.62581	28.31829
148.74871	85.73483	27.22760	28.23696
148.94474	86.04556	22.66047	28.22027
149.17414	86.54190	17.93068	28.30161
149.26590	87.08203	14.23944	28.38503
		11.72230	28.48096
		9.13009	28.56857
		7.17185	28.67907
		5.86011	28.77500
		5.20736	28.83340
		4.30645	28.83340
		3.42013	28.66239
		2.58178	28.32872
		1.82268	27.84281
		1.16785	27.22135
		0.64232	26.48935
		0.26485	25.66977
		0.04797	24.79597
		0.00000	23.89506

Pre-Processor for the Navier-Stokes Solver BTOB3D

Gridgen6.pas has been developed as a pre-processor for the code BTOB3D. Input files are produced in accordance with the BTOB3D users manual (Dawes, 1985). The input file contains geometrical and physical data. The physical data is required for the specification of boundary conditions, as discussed in Chapter 7, and for initial guesses to initiate the run. The geometrical data defines the domain and configuration of the computational mesh. As mentioned in Chapter 7, BTOB3D uses an H-mesh where the quasi-orthogonal and streamwise surfaces are surfaces of revolution. As mentioned in Chapter 6, it is necessary to construct a grid which clusters grid points in regions with high gradients and to have a smooth distribution in step sizes.

It is desirable to concentrate the grids near the leading edges and trailing edges of the blades to resolve the blade contours properly and also to have sufficient grid density in these high gradient regions. One method to establish leading and trailing-edge clustering is to use an elliptical distribution is shown in Figure 2C.1. A semi-ellipse is drawn such that the blade leading and trailing edges lie at the end of the ellipse (ie. x-intercept(s)). A radial line is then swept around the ellipse in equal angle increments of $\Delta\theta$, where $\Delta\theta=\pi/N$ and N is the number of grids in the blade passage between leading and trailing edges. Where the radial lines intersect the ellipse, a vertical line is dropped down to the x-axis, thus defining the coordinates of the grids. For the ellipse, the length of the semi-minor axis is a and the length of the semi-major axis is b as shown in Figure 2C.1. For length a fixed at unity, the length of b is used to control the shape of the ellipse. For larger values of b the changes in grid size near the leading and trailing edges will vary more rapidly. It was found that by using the elliptical distribution, clustering near leading and trailing edges was too rapid, exceeding the $\pm 30\%$ limit recommended by Dawes. Rapid clustering is caused by the infinite slope of the ellipse at the points where it intersects the x-axis. By shifting the ellipse down by an amount δ this problem was alleviated. The equation for the

3

OF/DE

3

PM-1 3½"x4" PHOTOGRAPHIC MICROCOPY TARGET
NBS 1010a ANSI/ISO #2 EQUIVALENT

Figure 2C.1: Elliptic clustering strategy.

ellipse is given by:

$$\frac{x^2}{a^2} + \frac{(y + \delta)^2}{b^2} = 1$$

and the equation of the radial line is

$$y = Mx$$

where $M = \tan(\theta)$ and $\theta_i = \theta_{i-1} + \Delta\theta$. Program *Ellipse.pas* was used to produce the grid spacing for grids 1 to 4 as outlined in Chapter 7. It was found that good results were achieved for spacing in the axial direction with the following values: $a=1$, $b=1.5$, and $\delta=0.65$. The same clustering procedure was used to determine the relative pitchwise spacing of the grids and good results were achieved with the following values: $a=1$, $b=1.62$, and $\delta=0.32$. Using these values produces a mesh very similar to that used by Isaacs (1993) with the pre-processor *Castle*, for trailing-edge damage studies.

Similarly to the grid configuration of Isaacs, the clustering upstream of the leading-edge plane and

downstream of the trailing-edge plane was determined by geometric progression. A geometric series is defined as follows:

$$a + ar + ar^2 + \dots + ar^{n-1} = \frac{a(1 - r^n)}{1 - r}$$

The relative spacing is defined by r and is fixed at a constant value (usually set at 1.15). The value of the x-coordinate nearest to the leading-edge or trailing-edge (defined as a) is set equal to the value of the adjacent node in the blade passage.

The problem with the elliptical clustering method was that the maximum relative step changes occurred in the vicinity of the leading and trailing edges and the minimum relative step changes occur midway between leading and trailing-edge planes. In regions of high gradients, the relative spacing should be low to ensure sufficient accuracy (Dawes, 1986b). Therefore, it makes little sense to have a scheme which gives maximum relative spacings in regions with highest gradients. Furthermore, by using a geometric progression upstream and downstream of the leading and trailing-edge planes, a constant and often high relative spacing was used. Due to memory constraints it was often difficult to sufficiently model the leading and trailing edges properly. A method should be devised which accounts for regions with high gradients by providing smaller grids with small relative step sizes. Such a method would produce small relative step sizes near the leading and trailing-edge planes, and larger relative spacing in regions with relatively small gradients such as in the vicinity of inflow and outflow boundaries as well as in the middle of the blade passage.

A clustering program, Cluster.pas, was devised to calculate suitable relative spacing values and then to determine the actual grid coordinates. Several functions were tested to produce suitable relative spacing distributions. The function finally chosen is defined as follows:

$$R_i = [(pi)^q + 1]^{(a+1)} + b$$

where

- R_i : is the relative spacing distribution
- i : is the node index (1..N)
- q : determines the general shape of the distribution (eg. quadratic or cubic)

- p: is the multiplying factor which controls the magnitudes of the R distribution
- α : $\cos [\pi/2 \cdot (i-1)/(N-1)]$
- $a\alpha+1$: is the damping factor which gives more weight to its argument at the beginning of the distribution (ie. leading and trailing edges) to change the relative spacing faster at lower values of i
- b: is used to shift the distribution: usually to "fine tune" the values at the beginning of the distribution

The above function can be used for the determination of the axial spacing in the blade passage or for the pitchwise relative spacing. The above-mentioned parameters are varied to get an appropriate (R) spacing distribution and to get a low value for the first grid relative to the trailing-edge or leading-edge radius. The number of grids and the range is fixed and specified. The distributions in these areas are symmetric and mirrored about mid-passage. For areas upstream of the leading edge and downstream of the trailing edge, the parameters must be adjusted to get the correct distances to the inflow and outflow boundaries respectively. The number of nodes is fixed and the coordinates of the first grid is fixed to be the same distance as the adjacent node in the blade passage. The following values were deemed satisfactory and were used for grid 5 ($i \times j \times k = 33 \times 103 \times 33$), as presented in Chapter 7:

x-distribution in the blade passage:

a=2. b=0.005. p=0.0001. q=0.5

x-distribution upstream of the leading-edge:

a=3.25. b=-0.025. p=0.0005. q=0.5

x-distribution downstream of the trailing-edge:

a=1.2. b=-0.04. p=0.00125. q=0.5

v-distribution:

a=2.0. b=0.0. p=0.0005. q=0.5

Future modifications to the scheme could include the possibility of un-symmetric distributions. This could be useful if larger gradients are expected on the suction side of the blade or if more clustering is desired at the trailing-edge and less at the leading-edge. Un-symmetric clustering is only necessary if the maximum number of nodes is restricted by the memory limitations of the computer.


```

.. This program is designed to produce input files for Tapes'
.. BROWID Mevies-sches solver. It is also capable of producing
.. CDL files for the production of mesh plots in CADREY.
.. The program is designed to run the CC2 blade geometry for a
.. linear cascade. Other blade profiles can be used by substituting
.. different files for CC2X.suc, CC2Y.suc, CC2X.pr, and CC2Y.pr.
.. Input files required are for: blade geometry, axial grid cooils,
.. circumferential and meridional relative spacing, and input b.c.'s.
.. The format of the output file produced is suitable for the current
.. version of 3Dflow.f
.. Typical values are hardwired into this version to avoid tedious
.. input for every run. See BROWID users manual for variable
.. definitions and card orders.

```

```

inc114 := 'CC2K.prs';
inc115 := 'CC2Y.prs';

(No. of data points in blade files)

sucmax := 45;
prmax := 59;
Xleeding := 0.0;
Xtrailing := 149.2659;
Metal_angle := 57.29;

Assign (infile, infill1);
Reset (infile);
  (Read in x values of grid nodes)
  For J := 1 to Xno do
    Readln (infile, X(J));
  Close (infile);

Assign (infile, infill2);
  (Read x-coord of blade suction surface)
  Reset (infile);
  For J := 1 to sucmax do
    Readln (infile, Xauc(J));
  Close (infile);

Assign (infile, infill3);
  (Read y-coord of blade suction surface)
  Reset (infile);
  For J := 1 to sucmax do
    Readln (infile, Yauc(J));
  Close (infile);

Assign (infile, infill4);
  (Read x-coord of blade pressure surface)
  Reset (infile);
  J := prmax;
Repeat
  Readln (infile, Xpre(J));
  J := J-1;
  Until (J = 0);
Close (infile);

Assign (infile, infill5);
  (Read y-coord of blade pressure surface)
  Reset (infile);
  J := prmax;
Repeat
  Readln (infile, Ypre(J));
  J := J-1;
  Until (J = 0);
Close (infile);

Circarc;
Write (' Enter total number of grid nodes in the circumf. direction (<= 33) : ');
Readln (Yno);
Write (' Enter total number of grid nodes in the spanwise direction (<= 33) : ');
Readln (Xno);
Write (' Enter file name for relative spacing in circumf. direction : ');
Readln (infill6);
Write (' Enter file name for relative spacing in spanwise direction : ');
Readln (infill7);

Assign (infile, infill6);
Reset (infile);
  For J := 1 to Yno-1 do
    Readln (infile, Yspacing(J));
  Close (infile);

Assign (infile, infill7);
Reset (infile);
  For K := 1 to Xno-1 do
    Readln (infile, Xspacing(K));
  Close (infile);

```

```

( Determining the number of grid nodes in the x direction that are
  inside the passage : )
J := 0;
Repeat
  J := J+1;
  Until (X(J) = X(leading));
J1 := J;

J := 0;
Repeat
  J := J+1;
  Until (X(J) = X(trailing));
J2 := J;

( Now, knowing the grid node axial locations inside the passage, determining
  the blade surface geometry at these axial locations by linear interpolation
  from the known blade surface points )
J := J1;
Repeat
  (Calculates suction surface y value at each J station)
  For I:= 1 to (smax-1) do
    If (X(J) >= Xsuc(I)) and (X(J) <= Xsuc(I+1)) then
      Y(J,2) := ((X(J)-Xsuc(I))/(Xsuc(I+1)-Xsuc(I)))*(Ysuc(I+1)) + Ysuc(I);
  (Calculates pressure surface y value at each J station)
  For I:= 1 to (pmax-1) do
    If (X(J) >= Xpr(I)) and (X(J) <= Xpr(I+1)) then
      Y(J,1) := ((X(J)-Xpr(I))/(Xpr(I+1)-Xpr(I)))*(Ypr(I+1)) + Ypr(I);
  J := J+1;
  Until (J > J2);

( Calculates y values from inlet to leading edge. )
For J := 1 to J1-1 do begin
  Y(J,1) := Y(J1,1);
  Y(J,2) := Y(J1,2);
end;

( Calculates y values from trailing edge to exit )
For J := J2 to Xno do begin
  Y(J,1) := Y(J2,1) + (X(J)-X(J2))*sin(Metal.Angle*PI/180.0)/
    cos(Metal.Angle*PI/180.0);
  Y(J,2) := Y(J2,2) + (X(J)-X(J2))*sin(Metal.Angle*PI/180.0)/
    cos(Metal.Angle*PI/180.0);
end;

Xno := Xno;
In := Yno;
Xn := Zno;

Clear;
Write (' Enter the Pitch of the cascade (110 71): ');
Readln (Pitch);
Nblade:=500;
Radius:=Pitch*Nblade/2/3.141593;

Write (' Enter the cascade span (mm) : ');
Readln (Span);
Write (' Enter the angle of the endwalls in deg. : ');
Readln (alpha);
(..... Variable hub/casing ..... )
for J:=1 to JN do begin
  R(J,1):=Radius-sin(alpha/57.29578)/cos(alpha/57.29578)*(X(J)-X(1));
  R(J,2):=sin(alpha/57.29578)/cos(alpha/57.29578)*(X(J)-X(1))*Radius*Span;
end;

Clear;
WriteIn('..... To create input file for Jflow.1 : Press 1 .....');
WriteIn('..... To create file for blade mesh: Press 2 .....');
WriteIn('..... To create file for meridional mesh: Press 3 .....');
WriteIn('.....');

```

```

Readln(cash);
If task = 1 then begin
  (..... )
  Now all the grid node location data necessary for input to Jflow.1 is given.
  Boundary condition information is now needed to be specified for input
  to Jflow.1
  (..... )
  Clear;
  Write (' Enter output file name to be used as input to Daves program : ');
  Readln (outfile);
  Assign (outfile, outfile);
  Rewrite (outfile);
  Write (' Enter the title of the current run. (max 18 characters) : ');
  Readln(Name);
  WriteIn (outfile, Name);
  Write (' Enter number of time steps to be marched : ');
  Readln (NMAX);
  JLE := 0;
  JTE := 2;
  NOBECT := 2; (see UTU3D users manual for defn.)
  INPUT := 2;
  INERT := 0;
  ( Card 2 : )
  WriteIn (outfile, IN:5, JN:5, NM:5, NBLADE:5, NMAX:5, JLE:5,
    JTE:5, NOBECT:5, INPUT:5, INERT:5);
  Clear;
  INMACH := 0;
  INVR := 0;
  INP := 0;
  IPOUT := 1;
  SMALL := 0;
  KERT := 0;
  ( Card 3 : )
  WriteIn (outfile, INMACH:5, INVR:5, INP:5, IPOUT:5, SMALL:5, KERT:5);
  (number of multigrid levels LVL (0 or 1))
  LVL := 1;
  N2H := 0;
  ( Card 4 : )
  WriteIn (outfile, LVL:5, N2H:5);
  ( Card 5 : )
  PMG:=2.0;
  WriteIn (outfile, PMG:10:5);
  ( Card 6 : )
  AV2H:=0.0;
  WriteIn (outfile, AV2H:10:5);
  For K := 1 to NOBECT do begin
    ( since the x coordinates, circumferential coordinates, blade thickness, and
      tailfe are all in mm, all multiplying factors are set to 0.001 )
    PAC1:=0.001; PAC2:=0.001; PAC3:=0.001; PAC4:=0.001;
    ( Card 7 : )
    WriteIn (outfile, PAC1:10:5);
    ( Card 8 : )
    J := 1;
    Repeat
      J:=J+1;

```

```

Repeat
  If J<Xno then begin
    write(Outfile,X(J):10:5);
    J:=J+1;
  end;
  I:=I+1;
  Until I=8;
  If J < (Xno+1) then begin
    write(Outfile,X(J):10:5);
    J:=J+1;
  end;
  Until J > Xno;

( Card 9 : )
  WriteIn (Outfile, PAC2:10:5);

( Card 10 : )
  Repeat
    I:=1;
  Until I=8;
  If J < (Xno+1) then begin
    write(Outfile,Y(J,1):10:5);
    J:=J+1;
  end;
  I:=I+1;
  Until I=8;
  If J < (Xno+1) then begin
    write(Outfile,Y(J,1):10:5);
    J:=J+1;
  end;
  Until J > Xno;

( Card 11 : )
  WriteIn (Outfile, PAC3:10:5);

( Card 12 : )
  J := 1;
  Repeat
    I:=1;
  Until I=8;
  If J<Xno then begin
    write(Outfile,Abs(Y(J,2)-Y(J,1)):10:5);
    J:=J+1;
  end;
  I:=I+1;
  Until I=8;
  If J < (Xno+1) then begin
    write(Outfile,Abs(Y(J,2)-Y(J,1)):10:5);
    J:=J+1;
  end;
  Until J > Xno;

( Card 13 : )
  WriteIn (Outfile, PAC4:10:5);

( Card 14 : )
  J := 1;
  Repeat
    I:=1;
  Until I=8;
  If J<Xno then begin
    write(Outfile,R(J,K):10:2);
    J:=J+1;
  end;
  I:=I+1;
  Until I=8;
  If J < (Xno+1) then begin
    write(Outfile,R(J,K):10:2);
    J:=J+1;
  end;
  Until J > Xno;

  Until J > Xno;
end;
(For 1 to NOSECT)
end;

Clear;
CP := 1005.4;
GA := 1.4;
WriteIn (' Enter value of time step multiplier ');
Write (' PT = 1/(1 + Mach.max); 0.5-0.75 is typical ');
Readln (PT);
WriteIn (' Enter Reynolds number based on axial chord ');
Write (' and exit meridional velocity ');
Readln (Re);
ELMAX:=10.0;
RPM := 0.0;
( Card 20 : )
  WriteIn (Outfile, CP:10:2,GA:10:2,PT:10:3,RE:10:2,ELMAX:10:3,RPM:10:3);

Clear;
Write (' Enter initial guess of static pressure on hub at inlet (Pa) ');
Readln (PUPHUB);
Write (' Enter initial guess of static pressure on tip at inlet (Pa) ');
Readln (PUPTIP);
Write (' Enter fixed value of static pressure on hub at exit (Pa) ');
Write (' Enter fixed value of static pressure on tip at exit (Pa) ');
Readln (POP);
( Card 21 : )
  WriteIn (Outfile, PUPHUB:10:3,PUPTIP:10:3,POHUB:10:3,POP:10:3);

Clear;
WriteIn (' Enter input file name for inlet boundary conditions ');
Write (' Pof (Pa), Tot (K), Vti (m/s), Vmi (m/s), ');
Write (' Swirl Angle (deg), Pitch Angle (deg) ');
Readln (Infil);
Assign (Infil, Infil);
Reset (Infil);
For K := 1 to KM do
  Readln (Infil, PO[K], TOT[K], VTIN[K], VMI[K], BR[K], BR[K]);
Close (Infil);
( Card 22, 23, 24, 25, 26 and 27 : )
  K := 1;
  Repeat
    I:=1;
  Until I=8;
  If K<KM then begin
    write(Outfile,PO[K]:10:2);
    K:=K+1;
  end;
  I:=I+1;
  Until I=8;
  If K < (KM+1) then begin
    write(Outfile,PO[K]:10:2);
    K:=K+1;
  end;
  Until K > KM;

  K := 1;
  Repeat
    I:=1;
  Until I=8;
  If K<KM then begin
    write(Outfile,TOT[K]:10:5);
    K:=K+1;
  end;
  I:=I+1;
  Until I=8;
  If K < (KM+1) then begin
    write(Outfile,TOT[K]:10:5);
    K:=K+1;
  end;
  Until K > KM;

  K := 1;
  Repeat
    I:=1;
  Until I=8;
  If K<KM then begin
    write(Outfile,VMI[K]:10:5);
    K:=K+1;
  end;
  I:=I+1;
  Until I=8;
  If K < (KM+1) then begin
    write(Outfile,VMI[K]:10:5);
    K:=K+1;
  end;
  Until K > KM;

  K := 1;
  Repeat
    I:=1;
  Until I=8;
  If K<KM then begin
    write(Outfile,VTIN[K]:10:5);
    K:=K+1;
  end;
  I:=I+1;
  Until I=8;
  If K < (KM+1) then begin
    write(Outfile,VTIN[K]:10:5);
    K:=K+1;
  end;
  Until K > KM;
end;

```

```

K := 1;
Repeat
  I:=1;
  Repeat
    if K<NM then begin
      write(Outfile,VTIME(K):10:5);
      K:=K+1;
    end;
    I:=I+1;
  Until I=8;
  If K < (NM+1) then begin
    write(Outfile,VTIME(K):10:5);
    K:=K+1;
  end;
Until K > NM;

K := 1;
Repeat
  I:=1;
  Repeat
    if K<NM then begin
      write(Outfile,VMI(K):10:5);
      K:=K+1;
    end;
    I:=I+1;
  Until I=8;
  If K < (NM+1) then begin
    write(Outfile,VMI(K):10:5);
    K:=K+1;
  end;
Until K > NM;

K := 1;
Repeat
  I:=1;
  Repeat
    if K<NM then begin
      write(Outfile,BS(K):10:5);
      K:=K+1;
    end;
    I:=I+1;
  Until I=8;
  If K < (NM+1) then begin
    write(Outfile,BS(K):10:5);
    K:=K+1;
  end;
Until K > NM;

K := 1;
Repeat
  I:=1;
  Repeat
    if K<NM then begin
      write(Outfile,BP(K):10:5);
      K:=K+1;
    end;
    I:=I+1;
  Until I=8;
  If K < (NM+1) then begin
    write(Outfile,BP(K):10:5);
    K:=K+1;
  end;
Until K > NM;

K := 1;
Repeat
  I:=1;
  Repeat
    if K<NM then begin
      write(Outfile,BM(K):10:5);
      K:=K+1;
    end;
    I:=I+1;
  Until I=8;
  If K < (NM+1) then begin
    write(Outfile,BM(K):10:5);
    K:=K+1;
  end;
Until K > NM;

K := 1;
Repeat
  I:=1;
  Repeat
    if K<NM then begin
      write(Outfile,Zspacing(K):10:5);
    end;
    I:=I+1;
  Until I=8;
  If K < (NM+1) then begin
    write(Outfile,Zspacing(K):10:5);
  end;
Until K > NM;

```

```

K:=K+1;
end;
I:=I+1;
Until I=8;
if K < NM then begin
  write(Outfile,Zspacing(K):10:5);
  K:=K+1;
end;
Until K > (NM-1);

[ Card 29 : ]
I := 1;
Repeat
  J:=1;
  Repeat
    if J<(IM-1) then begin
      write(Outfile,Yspacing(J):10:5);
      J:=J+1;
    end;
    J:=J+1;
  Until J=8;
  if I < IM then begin
    write(Outfile,Yspacing(I):10:5);
    I:=I+1;
  end;
  I:=I+1;
Until I > (IM-1);

[ Card 30 : ]
{sets the 5 time steps where results are sent to file}
write(Outfile,100:5, 500:5, 1000:5, 2000:5, 3000:5);

[ Card 31:12 : ]
For K := 1 to 11 do write (Outfile,K:2);
write(Outfile,12:2);
For K := 1 to 19 do write (Outfile,K:2);
write(Outfile,20:2);

[ Card 31 : ]
write(Outfile,1:5, 2:5, 3:5, 4:5, 5:5);

AV2 :=1.0;
AVA :=0.01;
AVA :=2.0;
[ Card 34 : ]
write(Outfile,1 0:10:5, 1.0:10:5, 1.0:10:5, AV2:10:5, AV4:10:5, AVA:10:5);

For K := 1 to 200 do
  TFAC[K]:=1.0;

MTIP :=NM;
[ Card 35 : ]
write(Outfile,NVIP:5,0:5,100,0:10:5);

[ Card 36 : ]
K := 1;
Repeat
  I:=1;
  Repeat
    if K<NM then begin
      write(Outfile,TFAC(K):10:5);
      K:=K+1;
    end;
    I:=I+1;
  Until I=8;
  if K < (NM+1) then begin
    write(Outfile,TFAC(K):10:5);
    K:=K+1;
  end;
Until K > NM;

```

```

Close (Outfile);
end; (Devs code output file)
(..... Calculating blade - blade and meridional .....
..... grid coordinates .....
.....
if task <= 1 then begin
Write('Enter name of grid coordinate output file. ');
Readln(Outfile2);
Assign(Outfile2, Outfile2); Rewrite(Outfile2);
end;
Yapsum:=0.0; Zapsum:=0.0;
for K:=1 to KM-1 do Yapsum:=Yapsum+Yapacing[K];
for K:=1 to KM-1 do Zapsum:=Zapsum+Zapacing[K];
(Calculating blade-blade mesh coordinates)
for J:=1 to JM do begin
YY(J,1):=Y(J,1);
for I:=2 to IM do begin
YY(I,1):=spacing[(I-1)/Yapsum*(Y(J,2)+PITCH Y(J,1))+YY(J,1-1)];
end;
end;
(Calculating meridional mesh coordinates)
for J:=1 to JM do begin
ZZ(J,1):=R(J,1)-Radius;
for K:=2 to KM do begin
ZZ(K,1):=Zapacing[(K-1)/Zapsum*(R(J,2)-R(J,1))+ZZ(J,K-1)];
end;
end;
if task = 2 then begin
(The following files have suitable format to be used as CDL files.)
for I:=1 to IM do begin
for J:=1 to JM do begin
WriteIn(Outfile2, 'POINT', X(J,1), Y(J,1), YY(I,1), ZZ(J,1));
end;
end;
I:=1; (sets level)
Write('Enter name of grid coordinate output file for the pressure surface. ');
WriteIn:
Readln(Outfile3);
Assign(Outfile3, Outfile3); Rewrite(Outfile3);
for J:=1 to JM do begin
WriteIn(Outfile3, 'POINT', X(J,1), Y(J,1), Pitch, ZZ(J,1));
end;
Write('Enter name of grid coordinate output file for the suction surface. ');
WriteIn:
Readln(Outfile4);
Assign(Outfile4, Outfile4); Rewrite(Outfile4);
for J:=1 to JM do begin
WriteIn(Outfile4, 'POINT', X(J,1), Y(J,2), YY(J,2), ZZ(J,1));
end;
end; (task 2)
if task = 3 then begin

```

PROGRAM ELLIPSE;

```

.....
** This program uses an ellipse to determine r's grid clustering.
** The ellipse is shifted in the negative y direction to reduce
** the clustering near the trailing edge. Upstream of the leading-
** edge and downstream of the trailing-edge a geometric distribution
** is used. Default values for CC2 blades are used as indicated.
**.....

```

```

Uses crt;

```

```

Type

```

```

index = -50..107;
vector1 = array [index] of real;

```

```

Var
  x, dx, theta, change, x2, y2
  : vector1;
  a, b, M, Scale, Scale2
  : real;
  xle, xte, xinter, dy, AA, BB, CC
  : real;
  i, a2, b2, dy2
  : integer;
  M, NLE, RTE, k, half, Ymax,
  : string;
  st1, st2
  : text;
  file1, file2
  : file;

```

```

Begin

```

```

  clrscr;
  write ('Enter the name of the data file to contain the x-coords. ');
  readln (st1);
  assign (file2, st1);
  rewrite (file2);
  write ('Enter the name of the data file to contain y rel. spacing: ');
  readln (st1);
  assign (file1, st1);
  rewrite (file1);

```

```

  write ('Enter number of total nodes in x-direction: ');
  readln (M);
  write ('Enter the node index of the leading edge: ');
  readln (JLE);
  write ('Enter the node index of the trailing edge: ');
  readln (JTE);
  M:=JTE-JLE;

```

```

  half := trunc(M/2);
  pi:= 4.0*arctan(1.0);
  dtheta := pi/M;
  xle := 0;
  xte := 149.2659; {values for C2 blades}
  x[0] := xle;
  x[M] := xte;
  x[half] := (xte-xle)/2;
  theta[0] := 0.0;
  a := 1.0; {semi x axis length}

```

```

  clrscr;
  write ('Enter semi-major axis length (b) for x clustering. ');
  readln (b); {semi y axis length}
  write ('Enter the ellipse shift value (delta=0) for x clustering. ');
  readln (dy); {shift of ellipse in the neg. y dir}
  xinter := sqrt(a*(1-sqrt(b/b2))); {x intercept of ellipse}
  scale := (xte-xle)/2/xinter;

```

```

{Calculation of blade passage grid spacing}

```

```

for i := 1 to half-1 do begin
  theta[i] := theta[i-1] + dtheta;
  M := sin(theta[i])/cos(theta[i]);
  AA := 1/a*(1+M*b/b2);
  BB := 2*b*dy/b2/b;
  CC := dy*dy/b2/b;
  x[i] := -BB/2/AA + sqrt(BB*BB - 4*AA*CC)/2/AA;
end;

```

```

for i := 1 to half-1 do begin
  x[i] := xinter - x[i];
  x[i] := scale * x[i];
end;
{symmetric clustering for other half of blade passage}
k := 1;
for i := half+1 to M-1 do begin
  x[i] := xte - x[half-k];
  k := k + 1;
end;

```

```

{calculating grid locations upstream of the leading edge.}

```

```

clrscr;
write ('Enter the spacing factor (r) for upstream of the leading edge: ');
readln (r);
NLE := 1-JLE;
x[-1] := -x[i];
for i := 2 downto NLE do begin
  x[i] := (-x[-1])*(1-exp(-1*ln(r)))/(1-r);
end;

```

```

{calculating grid locations downstream of trailing edge.}

```

```

write ('Enter the spacing factor (r) for downstream of the trailing edge: ');
readln (r);
RTE := M - JTE;
x[M+1] := x[M] * x[i];
for i := M+2 to RTE do begin
  x[i] := (x[M+1]-x[M])*(1-exp((1-M)*ln(r)))/(1-r) + x[M];
end;

```

```

{calculate relative change}

```

```

for i := NLE to RTE-2 do begin
  change[i] := (x[i+2]-x[i+1])/x[i+1];
end;

```

```

clrscr;

```

```

for i := NLE to RTE do begin
  writeln(file2, x[i]:10:5);
  writeln(x[i]:10:5, change[i]:0:3);
end;
readln;

```

```

{Calculation of blade to blade spacing. i.e. y spacing}

```

```

write ('Enter the number of nodes in the pitchwise direction: ');
readln (Ymax);
half := trunc((Ymax-1)/2);
dtheta := pi/(Ymax-1);
a2 := 1.0; {semi x axis length}
write ('Enter the ellipse semi-major axis length (b1): ');
readln (b2); {semi y axis length}
write ('Enter the ellipse shift factor (delta=0) ');
readln (dy2);

```

```

xinter := sqrt(a2*(1-sqrt(dy2/b2))); {x intercept of ellipse}

```

```

for i := 1 to half-1 do begin
  theta[i] := theta[i-1] + dtheta;
  M := sin(theta[i])/cos(theta[i]);
  AA := 1/a2*(1+M*b2/b2);
  BB := 2*b2*dy2/b2/b2;
  CC := dy2*dy2/b2/b2;
  x2[i] := -BB/2/AA + sqrt(BB*BB - 4*AA*CC)/2/AA;
end;

```

```

x2[half] := 0.0;

```

```

scale := 1 / (xinter-x2[i]);
for i := 1 to half do begin
  x2[i] := xinter - x2[i];
  x2[i] := scale * x2[i];
end;

```

```

y2(0) := 0.0; y2(1) := 1.0;
for i := 2 to half do begin
  y2(i) := (x2(i-1)-x2(1))/(x2(1)-x2(0));
end;

k := 0;
for i := half+1 to ymax-1 do begin
  y2(i) := y2(half-k);
  k := k + 1;
end;

for i := 1 to half-2 do begin
  change(i) := y2(i+1)/y2(1);
end;

direct:
for i := 1 to ymax-1 do begin
  writeln(file1,y2(i):10:5);
end;
for i := 1 to half do begin
  writeln(y2(i):10:5,change(i):10:3);
end;
readln;
close(file1);
close(file2);
end.

```

PROGRAM CLUSTER;

```

(*.....*)
(* This program creates files containing x-coordinate grid data *)
(* and y relative spacing data for input to CRIZM2D.pas. The *)
(* program is intended to be used for CC2 blades but can easily *)
(* be modified for any other blade geometry. *)
(*.....*)

Uses crt;

type index1 = -30..107;
vector = array[index1] of real;

Var
  i,j,k,M,Npas,M1,Mt,M1,Mhalf,JM,JLS,JTS integer;
  at1,at2 setting;
  file1,file2 text;
  R,M,M1,M1,xrel1 vector;
  p,q,a,alpha,b scale,f;
  pl,q1,a1,b1,f1;
  pr,qr,at,bt,ft;
  pl,q1,a1,b1,f1,scale1 real;

Begin
  clrscr;
  writeln(' *** Produce x-coordinate data *** ');
  write('Enter name of output file of x-coordinates: ');
  readln(at1);
  assign(file1,at1);
  rewrite(file1);
  write('Enter the total number of (odd) nodes in the x-direction: ');
  readln(JM);
  write('Enter the (even) node index of the leading-edge: ');
  readln(JLS);
  write('Enter the (even) node index of the trailing-edge: ');
  readln(JTS);
  clrscr;
  writeln('Compute grid clustering in the blade passage. ');
  write('Enter p: ');
  readln(p);
  write('Enter q: ');
  readln(q);
  write('Enter a: ');
  readln(a);
  write('Enter b: ');
  readln(b);
  N:=trunc((JTS-JLS)/2);
  passage:
  for i:=1 to N do begin
    alpha:=cos(3.1415927/2*(i-1)/2/(N-1));
    R(i):=exp((a*alpha+1)*ln( exp(q*ln(p*1))+1 ))*b;
    if i = 15 then readln;
  end;

  x(0):=0.0;
  x(1):=1.0;
  for i:=2 to N do begin
    f:=1.0;
    for j:=1 to i-1 do begin
      f:=f*R(j);
    end;
    x(i):=(x(1)-x(0))*f*x(i-1);
  end;
  scale:=149.2659/2/(x(N)-x(0));
  for i:=1 to N do begin
    x(i):=scale*x(i);
  end;

```

```

( for i:=Npas to Mt do begin
  writeln(i:10,R[i]:15,S,X[i]:15,S);
  if i = 65 then readln;
end;
readln;
)

writeln;
writeln ('      i      x(i):');
for i:=1 to Mt do begin
  writeln(i:10,X[i]:10,S);
  writeln(file1,X[i]:10,S);
  if i = 0 then readln;
  if i = 10 then readln;
end;
readln;

(*****Calculating pitchwise relative spacing*****
clear:
writeln (' *** Produce pitchwise relative spacing *** ');
writeln;
write ('Enter the output file name: ');
readln (st2);
assign(file2,st2);
rewrite(file2);

writeln; writeln;
writeln ('Compute grid clustering in pitchwise direction. ');
writeln;
write ('Enter p: ');
readln (pt);
write ('Enter q: ');
readln (qt);
write ('Enter a: ');
readln (at);
write ('Enter b: ');
readln (bt);
write ('Enter number of (odd) nodes in pitchwise direction: ');
readln (N1);
N1:=N1-1;      (number of pitchwise cells in the blade passage)
N1half:=trunc(N1/2);

for i:=1 to trunc(N1/2) do begin
  alpha:=cos(3.1415927/2*((i-1)/2/(N1-1)));
  R[i]:=exp((at*alpha+1)*ln( exp(qt*ln(pt*i)+1 ))*bt);
  if i = 15 then readln;
end;

x1[0]:=0.0;
x1[1]:=x1.0;
for i:=2 to trunc(N1/2) do begin
  x1:=1.0;
  for j:=1 to i-1 do begin
    x1:=x1*x1(i);
  end;
  x1[i]:=(x1[1]-x1[0])*(x1[0])+(x1[i-1]);
end;

scale:=110.71/2/(x1[N1half]-x1[0]);
for i:=1 to trunc(N1/2) do begin
  x1[i]:=x1[i]*scale;
end;

for i:=1 to trunc(N1/2) do begin
  x1[i]:=(x1[i]-x1[1-1])/(x1[i]-x1[0]);
end;

for i:=trunc(N1/2)+1 to N1 do begin
  x1[i]:=x1[i];
end;

```


Sample Input File: Transonic, Parallel Endwall

```

xrell[i] := xrell[i];
h: h = h - 1;
end;

for i = 1 to trunc(M1/2) do begin
    write(h, 10, M1[i], 10, 4, x[i], 15, 5, xrell[i], 15, 5);
    if i = trunc(M1/2) then readln;
end;

)

writeln; writeln;
for i = 1 to M1 do begin
    write(xrell[i], 10, 5);
    write(file2, xrell[i], 10, 5);
    if i = M1 then readln;
end;

readln;
close(file1);
close(file2);

end.

```

[illegible][illegible]

- 184 -

References

Ainley, D.G., and Mathieson, G.C.R., "A Method of Performance Estimation for Axial Flow Turbines", ARC, R&M 2974, 1951.

Anderson, D.A., Tannehill, J.C., Pletcher, R.H., 1984, "Computational Fluid Mechanics and Heat Transfer", Hemisphere Publishing Corporation, New York.

Amone, A., Swanson, R.C., 1993, "A Navier-Stokes Solver for Turbomachinery Applications", ASME J. Turbo., Vol.115, pp.305-313.

Baghdadi, Samy, 1977, "The Effect of Rotor Blade Wakes on Centrifugal Compressor Diffuser Performance - A Comparative Experiment", ASME J. Fluids Eng., March, 1977, pp.45-52.

Baldwin, B., Lomax, H., 1978, "Thin-Layer Approximation and Algebraic Model for Separated Turbulent Flows", AIAA paper 78-257.

Barber, Thomas J., 1987, "Computational Fluid Dynamics for Internal Flows", AIAA J. Propulsion, Vol.3, No.5, pp.385-386.

Beam, R.M., Warming, R.F., 1978, "An Implicit Factored Scheme for the Compressible Navier-Stokes Equations", AIAA J., Vol.16, No.4, pp.393-402.

Blair, L.W., and Russo, C.J., 1980, "Compact Diffusers for Centrifugal Compressors", AIAA paper 80-1077.

Bradley, C.I., and Cockrell, D.J., 1971, "The Response of Diffusers to Flow Conditions at Their Inlet", Paper 5, Symposium on Internal Flows, Univ. of Salford, April 20-22, 1971, Proc. pp.A32-A41.

Brandt, A., 1977, "Multi-Level Adaptive Solutions to Boundary-Value Problems", Math. Comp., Vol.31, No.138, pp.333-390.

Brandt, A., 1982, "Introductory Remarks on Multigrid Methods", Numerical Methods for Fluid Dynamics, Academic Press, New York.

Briley, W.R., McDonald, H., 1977, "Solution of the Multidimensional Compressible Navier-Stokes Equations by a Generalized Implicit Method", J. Comp. Physics, Vol.24, No.4, p.372.

Brown, L.E., 1972, "Axial Flow Compressor and Turbine Loss Coefficients: A Comparison of Several Parameters", ASME J. Eng. for Power, July, 1972.

Chima, Rodrick V., 1987, "Explicit Multigrid Algorithm for Quasi-Three-Dimensional Viscous Flows in Turbomachinery", AIAA J. Propulsion, Vol.3, No.5, pp.397-405.

Chima, Rodrick V., 1991, "Viscous Three-Dimensional Calculations of Transonic Fan Performance", AGARD-CP-510, paper 21.

Chmielewski, G.E., 1974, "Boundary-Layer Considerations in the Design of Aerodynamic Contractions", AIAA J. Aircraft, Vol.11, No.8, pp.435-438.

Chorin, A.J., 1967, "A Numerical Method for Solving Incompressible Viscous Flow Problems", J. Comp. Physics, Vol.2, pp.12-26.

Chung, H., Baek, J., 1992, "Influence of Trailing-Edge Grid Structure on Navier-Stokes Computation of Turbomachinery Cascade Flow", Int. J. Num. Meth. in Fluids, Vol.15, pp.883-894.

Cleak, J.G.E., Gregory-Smith, D.G., 1992, "Turbulence Modelling for Secondary Flow Prediction in a Turbine Cascade", ASME J. Turbo., Vol.114, pp.590-598.

Cumpsty, N.A., 1989, "Compressor Aerodynamics", Longman, England.

Davis, R.L., Ni, R., Carter, J.E., 1987, "Cascade Viscous Flow Analysis Using the Navier-Stokes Equations", AIAA J. Propulsion, Vol.3, No.5, pp.406-413.

Dawes, W.N., 1985, "BTOB3D: A Computer Program for the Analysis of Three-Dimensional Viscous Compressible Flow in Turbomachinery Blade Rows", BTOB3D Users Manual.

Dawes, W.N., 1986a, "A Numerical Method for the Analysis of 3D Viscous Compressible Flow in Turbine Cascades; Application to Secondary Flow Development in a Cascade With and Without Dihedral", ASME paper 86-GT-145.

Dawes, W.N., 1986b, "Numerical Techniques for Viscous Flow Calculations in Turbomachinery Bladings; Application of Full Navier-Stokes Solvers to Turbomachinery Flow Problems", VKI LS 1986-02.

Dawes, W.N., 1986c, "A Pre-Processed Implicit Algorithm for 3D Viscous Compressible Flow", Proc. 6th GAMM Conf. Num. Meth. Fluid Mech., Vieweg, pp.70-77.

Dawes, W.N., 1987, "A Numerical Analysis of the Three-Dimensional Viscous Flow in a Transonic Compressor Rotor and Comparison With Experiment", ASME J. Turbo., Vol.109, pp.83-90.

Dawes, W.N., 1988, "Development of a 3D Navier Stokes Solver for Application to all Types of Turbomachinery", ASME paper 88-GT-70.

Dawes, W.N., 1990, "A Comparison of Zero and One Equation Turbulence Modelling for Turbomachinery Calculations", ASME paper 90-GT-303.

Dawes, W.N., 1992, "The Simulation of Three-Dimensional Viscous Flow in Turbomachinery Geometries Using a Solution-Adaptive Unstructured Mesh Methodology", ASME J. Turbo., Vol.114, pp.528-537.

Dawes, W.N., 1993, "The Extension of a Solution-Adaptive Three-Dimensional Navier-Stokes Solver Toward Geometries of Arbitrary Complexity", ASME J. Turbo., Vol.115, pp.283-295.

Dean, R.C., 1971, "On the Unresolved Fluid Dynamics of the Centrifugal Compressor", In Dean R C (ed), Advanced Centrifugal Compressors, ASME, ch1.

Dean, R.C., and Senoo, Y., 1960, "Rotating Wakes in Vaneless Diffusers", ASME J. Basic Eng., Sept. 1960, pp.563-574.

Denton, J.D., 1993, "Loss Mechanisms in Turbomachines", ASME paper 93-GT-435.

Denton, J.D., and Cumpsty, N.A., 1987, "Loss Mechanisms in Turbomachines", Proc. I. Mech E., 1987-6, "Turbomachinery - Efficiency Prediction and Improvement", pp.1-14.

Denton, J.D., Xu, L., 1990, "The Trailing Edge Loss of Transonic Turbine Blades", ASME J. Turbo., Vol.112, pp.277-285.

Dorney, D.J., Davis, R.L., 1991, "Navier-Stokes Analysis of Turbine Blade Heat Transfer and Performance", AGARD-CP-510, paper 22.

Dunham, J., 1974, "The Effect of Stream Surface Convergence on Turbomachine Boundary Layers", Aeronautical J., Vol.78, No.758/759, pp.90-92.

Eckardt, D., 1975, "Instantaneous Measurements in the Jet-Wake Discharge Flow of a Centrifugal Compressor Impeller", ASME J. Eng. for Power, July, 1975, pp.337-346.

Eckardt, D., 1979, "Jet-wake mixing in the diffuser entry region of a high speed centrifugal compressor.", Joint Symposium on Design and Operation of Fluid Machinery 1:301-20 IAHR/ASME/ASCE

Fletcher, C.A.J., 1988a, "Computational Techniques for Fluid Dynamics. Volume I: Fundamental and General Techniques", Springer-Verlag, New York.

Fletcher, C.A.J., 1988, "Computational Techniques for Fluid Dynamics. Volume II: Specific Techniques for Different Flow Categories", Springer-Verlag, New York.

Fottner, L., 1972, "Analytical Approach for the Loss and Deflection Behaviour of Cascades in Transonic Flow Including Axial Mass Flow Variation", AGARD-AG-164, paper 1-7, pp.119-139.

Galpatti, R., Hawthorne, W.R., 1978, "Cascades with Arbitrary End Wall Contraction", ARC RM No. 3813.

Goobie, Scott Malcolm, 1989, "*A Test Section for Investigating the Off-Design Performance of Planar Cascades*", Master's Thesis, Carleton University.

Gostelow, J.P., 1984, "*Cascade Aerodynamics*", Pergamon Press, Oxford.

Graham, C. G., and Kost, F.H., 1979, "*Shock Boundary Layer Interaction on High Turning Transonic Turbine Cascades*", ASME paper 79-GT-37.

Granville, Paul S., 1987, "*Baldwin-Lomax Factors for Turbulent Boundary Layers in Pressure Gradients*", AIAA J. Technical Notes, Vol.25, No.12, pp.1624-1627.

Haines, A.B., 1982, "*Turbulence Modelling*", Aeronautical J., pp.269-277.

Hänel, D., 1991, "*Computational Techniques for Solving the Navier-Stokes Equations*", AGARD-CP-510, paper 1.

Heilmann, W., 1972, "*The Influence of Axial Velocity Density Ratio on Compressor Cascade Performance in Compressible Flow*", AGARD-AG-164, paper I-13, pp.221-240.

Henne, P.A., 1990, "*Applied Computational Aerodynamics*", Vol.125 Progress in Astronautics and Aeronautics, AIAA, Washington D.C.

Hieu, Minh, 1992, "*The Impact of Turbulence Modelling on the Numerical Predictions of Flows*", Lecture Notes in Physics: 414, Springer-Verlag, New York.

Hirsch, C., 1985, "*An Analysis of Flow Models: From Navier-Stokes to Potential Flow Equations*", AGARD-LS-140, paper 2.

Hoffmann, J.A., 1981, "*Effects of Free-Stream Turbulence on Diffuser Performance*", ASME J. Fluids Eng., Vol.103, pp.385-390.

Hoffmann, Klaus A., 1989, "*Computational Fluid Dynamics for Engineers*", Engineering Education System, Austin, Texas.

Humphrey, J.A.C., Whitelaw, J.H., Yee, G., 1981, "*Turbulent Flow in a Square Duct with Strong Curvature*", J. Fluid Mech., Vol.103, pp.443-463.

Ilyas, M., Norbury, J.F., 1973, "*Effect of Axial Velocity Variation on the Subsonic Flow Through a Compressor Cascade*", Inst. Mech. Engr. Conf. Publ. 3., paper C112/73, pp.276-287.

Inoue, M., and Cumpsty, N.A., 1984, "*Experimental Study of Centrifugal Impeller Discharge Flow in Vaneless and Vaned Diffusers*", ASME J. Eng. for Gas Turb. and Power, Vol.106, pp.455-467.

Isaacs, David, 1993, "*Aerodynamics of Turbine Blades with Trailing-Edge Damage: Measurements and Computations*", Master's Thesis, Carleton University.

Jameson, A., 1983, "*The Evolution of Computational Methods in Aerodynamics*", ASME J. Appl. Mech., Vol.50, pp.1052-1070.

Jameson, A., Baker, T.J., 1983, "*Solution of the Euler Equations for Complex Configurations*", AIAA paper 83-1929.

Jameson, A., Baker, T.J., 1984, "*Multigrid Solution of the Euler Equations for Aircraft Configurations*", AIAA paper 84-0093.

Jameson, A., Yoon, S., 1986, "*Multigrid Solution of the Euler Equations Using Implicit Schemes*", AIAA J., Vol.24, No.11, pp.1737-1743.

Johnson, Gary M., 1982, "*Relaxation Solutions of the Full Euler Equations*", Lecture Notes in Physics 170, Springer-Verlag, New York.

Karniadakis, G.E., Orszag, S.A., 1993, "*Nodes, Modes and Flow Codes*", Physics Today, March, pp.34-42.

Kenny, D.P., 1968, "*A Novel Low Cost Diffuser for High Performance Centrifugal Compressors*", ASME paper 68-GT-38.

Kenny, D.P., 1972, "*A Comparison of the Predicted and Measured Performance of High Pressure Ratio Centrifugal Compressor Diffusers*", ASME paper 72-GT-54.

Kiock, R., Lehthaus, F., Baines, N.C., Sieverding, C.H., 1986, "*The Transonic Flow Through a Plane Turbine Cascade as Measured in Four European Wind Tunnels*", ASME J. Eng. for Gas Turb. and Power, Vol.108, pp.277-284.

Klein, A., 1981, "*REVIEW: Effects of Inlet Conditions on Conical-Diffuser Performance*", ASME J. Fluids Eng., Vol.103, pp.250-257.

Klein, Manfred P., 1989, "*Measurements of Total Pressure Losses in a Conical Diffuser*", Master's Thesis, Carleton University.

Kubota, S., 1962, "*Cascade Performance with Accelerated or Decelerated Axial Velocity*", Bulletin of JSME, Vol.5, No.19, pp.450-460.

Langston, L.S., Nice, M.L., Hooper, R.M., "*Three-Dimensional Flow Within a Turbine Cascade Passage*", ASME J. Eng. for Power, pp.21-28.

Lumley, J.L., 1989, "*Whither Turbulence? Turbulence at the Crossroads*", Lecture Notes in Physics: 357, Springer-Verlag, New York.

Lyman, F.A., 1993, "*On the Conservation of Rothalpy in Turbomachines*", ASME J. Turbo., Vol.115, No.3, pp. 520-526.

MacCormack, R.W., 1969, "*The Effect of Viscosity in Hypervelocity Impact Cratering*", AIAA Paper 69-354.

MacCormack, R.W., 1982, "*A Numerical Method for Solving the Equations of Compressible Viscous Flow*", AIAA J., Vol.20, No.9, pp.1275-1281.

- McNally, W.D., Sockol, P.M., 1985, "Review - Computational Methods for Internal Flows with Emphasis on Turbomachinery", ASME J. Fluids Eng., Vol.107, pp.6-22.
- Marvin, Joseph G., 1983, "Turbulence Modelling for Computational Aerodynamics", AIAA J., Vol.21, No.7, pp.941-955.
- Masek, E.H., Norbury, J.F., 1973, "Low-Speed Performance of a Compressor Cascade Designed for Prescribed Velocity Distribution and Tested with Variable Axial Velocity Ratio", Inst. Mech. Engr. Conf. Publ.3, paper C88/73, pp. 224-236.
- Mee, D.J., Baines, N.C., Oldfield, M.L.G., Dickens, T.E., 1992, "An Examination of the Contributions to Loss on a Transonic Turbine Blade in Cascade", ASME J. Turbo., Vol.114, pp.155-162.
- Mikhail, M.N., Rainbird, W.J., 1978, "Optimum Design of Wind Tunnel Contractions", AIAA paper 78-819.
- Miller, Donald S., 1978, "Internal Flow Systems", Vol.5 in BHRA Fluid Engineering Series, U.K.
- Morel, T., 1975, "Comprehensive Design of Axisymmetric Wind Tunnel Contractions", ASME J. Fluid Eng., June, 1975, pp.225-233.
- Nakahashi, K., Nozaki, O., Kikuchi, K., Tamura, A., 1989, "Navier-Stokes Computations of Two and Three-Dimensional Cascade Flowfields", AIAA J. Propulsion, Vol.5, No.3, pp.320-326.
- Nakajima, K., Kallinderis, Y., 1994, "Comparison of Finite Element and Finite Volume Methods for Incompressible Viscous Flows", AIAA J., Vol.32, No.5, Technical Notes, pp.1090-1093.
- Nituch, M.J., Sjolander, S., Head, M.R., 1978, "An Improved Version of the Cebeci-Smith Eddy-Viscosity Model", Aeronautical Quarterly, pp.207-225.
- Ohuonu, Z., Shaw, R., 1973, "Effect of Axial Velocity Variation on Deviation for Compressor Cascades", Inst. Mech. Engr. Conf. Publ. 3, paper C53/73, pp.110-114.
- Patankar, S.V., Spalding, D.B., 1972, "Heat and Mass Transfer in Boundary Layers", 2nd Ed., Intertext Books, London.
- Perdichizzi, A., 1990, "Mach number Effects on Secondary Flow Development Downstream of a Turbine Cascade", ASME J. Turbo., Vol.112, pp.643-651.
- Pollard, D., Costelow, J.P., 1967, "Some Experiments at Low Speed on Compressor Cascades", ASME J. Eng. for Power, Vol.89, pp.427-436.
- Pollard, D., Horlock, J.H., 1963, "A Theoretical Investigation of the Effect of Change in Axial Velocity on the Potential Flow Through a Cascade of Airfoils", ARC CP 619.
- Raw, M.R., Galpin, P.F., Hutchinson, B.R., 1989, "Co-Located Finite Volume Method for Solving the Navier-Stokes Equations for Incompressible and Compressible Flows in Turbomachinery: Results and Applications", Can. Aero. & Space J., Vol.35, No.4, pp.189-196.

Reneau, L.R., Johnston, J.P., Kline, S.J., "*Performance and Design of Straight, Two-Dimensional Diffusers*", ASME J. Basic Eng., March 1967, pp.141-150.

Rieger, H., Jameson, A., "*Solution of Steady Three-Dimensional Compressible Euler and Navier-Stokes Equations by an Implicit LU Scheme*", AIAA paper 88-0619.

Rodi, W., 1982, "*Examples of Turbulence Models for Incompressible Flows*", AIAA J., Vol.20, No.7, pp.872-879.

Rodger, Philippe, 1992, "*Establishing Two-Dimensional Flow in a Large-Scale Planar Turbine Cascade*", Master's Thesis, Carleton University.

Rodger, P., Sjolander, S.A., Moustapha, S.H., 1992, "*Establishing Two-Dimensional Flow in a Large-Scale Planar Turbine Cascade*", AIAA paper 92-3066.

Rojas, J., Whitelaw, J.H., and Yianneskis, M., 1987, "*Forced Convective Heat Transfer in Curved Diffusers*", ASME J. Heat Transfer, Vol.109, pp.866-871.

Sharma, O.P., Butler, T.L., 1987, "*Predictions of Endwall Losses and Secondary Flows in Axial Flow Turbine Cascades*", ASME J. Turbo., Vol.109, pp.229-236.

Sharma, O.P., Graziani, P.A., 1983, "*Influence of Endwall Flow on Airfoil Suction Surface Midheight Boundary Layer Development in a Turbine Cascade*", ASME J. Eng. for Power, Vol.105, pp.147-155.

Siden, G.L.D., Dawes, W.N., Albråten, P.J., 1990, "*Numerical Simulation of the Two-Dimensional Viscous Compressible Flow in Blade Cascades Using a Solution-Adaptive Unstructured Mesh*", ASME J. Turbo., Vol.112, pp.311-319.

Sieverding, C.H., 1985a, "*Secondary Flows in Straight and Annular Turbine Cascades*", Thermodynamics and Fluid Mechanics of Turbomachinery, Vol. II, Proceedings of NATO Advanced Study Institute, 1985.

Sieverding, C.H., 1985b, "*Recent Progress in the Understanding of Basic Aspects of Secondary Flows in Turbine Blade Passages*", ASME J. Eng. for Gas Turb. and Power, Vol.107, pp.248-257.

Sjolander, S.A., 1993, "*Course Notes: Viscous Flows*", Dept. Mech. and Aero. Eng., Carleton University, Ottawa.

Sjolander, S.A., 1994, "*CFD Code Assessment in Turbomachinery*", Notes from Forum held at ASME-IGTI Turbo Expo, The Hague, June 16, 1994.

Sjolander, S.A., Isaacs, D., Klein, W.A., 1993, "*Aerodynamics of Turbine Blades With Trailing-Edge Damage: Measurements and Computations*", To be presented at XI Int. Symp. on Air Breathing Engines, Sept.93, Tokyo, Japan.

Sjolander, S.A., and Saravanamuttoo, H.I.H., 1989, "*Gas Turbine and Turbomachinery Education at Carleton University*", ASME paper 89-GT-16.

Speziale, C.G., Abid, R., Anderson, E.C., 1992, "Critical Evaluation of Two-Equation Models for Near-Wall Turbulence", AIAA J., Vol.30, No.2, pp.324-331.

Stark, U., 1972, "The Effect of Axial Velocity Ratio on the Aerodynamic Coefficients of a Compressor Cascade in Viscous Flow", AGARD-AG-164, paper I-12, pp.205-220.

Stark, U., Hoheisel, H., 1981, "The Combined Effect of Axial Velocity Density Ratio and Aspect Ratio on Compressor Cascade Performance", ASME J. Eng. for Power, Vol.103, pp.247-255.

Starke, J., 1981, "The Effect of Axial Velocity Density Ratio on the Aerodynamic Coefficients of Compressor Cascades", ASME J. Eng. for Power, Vol.103, pp.210-219.

Starken, H., Breugelmanns, F.A.E., Schimming, P., 1975, "Investigation of the Axial Velocity Density Ratio in a High Turning Cascade", ASME paper 75-GT-25.

Stevens, S.J., and Williams, G.J., "The Influence of Inlet Conditions on the Performance of Annular Diffusers", ASME J. Fluids Eng., Vol.102, pp.357-363.

Storer, J.A., Cumpsty, N.A., 1993, "An Approximate Analysis and Prediction Method for Tip Clearance Loss in Axial Compressors", ASME paper 93-GT-140.

Stratford, B.S., 1959, "The prediction of separation of the turbulent boundary layer", J.Fluid Mech., Vol.5, pp.1-16.

Swanson, R.C., Turkel, E., 1987, "Artificial Dissipation and Central Difference Schemes for the Euler and Navier-Stokes Equations", AIAA paper 87-1107.

Taylor, A.M.K.P., Whitelaw, J.H., and Yianneskis, M., 1982, "Curved Ducts with Strong Secondary Motion: Velocity Measurements of Developing Laminar and Turbulent Flow", ASME J. Fluids Eng., Vol.104, pp.350-359.

Thompson, Joe F., 1984, "Grid Generation Techniques in Computational Fluid Dynamics", AIAA J., Vol.22, No.11, pp.1505-1523.

Thompson, J.F., Warsi, Z.U., Mastin, C.W., 1982, "Boundary-Fitted Coordinate Systems for Numerical Solution of Partial Differential Equations - A Review", J. Comp. Physics, Vol.47, pp.1-108.

Tindell, R.H., 1987, "Highly Compact Inlet Diffuser Technology", AIAA paper 87-1747.

Tremblay, B., 1989, "Measurement and Correlation of Off-Design Performance of Axial-Flow Turbines", Master's Thesis, Carleton University.

Tremblay, B., Sjolander, S.A., and Moustapha, S.H., 1990, "Off-Design Performance of a Linear Cascade of Turbine Blades", ASME paper 90-GT-314.

Turner, M.G., Jennions, I.K., 1993, "An Investigation of Turbulence Modelling in Transonic Fans Including a Novel Implementation of an Implicit k-

END

2 2-1 1-9 6

FIN