1

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

PRECISIONSM RESOLUTION TARGETS

National Library
of Canada

Bibliothèque nationale
du Canada

Acquisitions and
Bibliographic Services Branch

Direction des acquisitions et
des services bibliographiques

395 Wellington Street
Ottawa, Ontario
K1A 0N4

395, rue Wellington
Ottawa (Ontario)
K1A 0N4

Your file Votre référence

Our file Notre référence

NOTICE

The quality of this microform is heavily dependent upon the quality of the original thesis submitted for microfilming. Every effort has been made to ensure the highest quality of reproduction possible.

If pages are missing, contact the university which granted the degree.

Some pages may have indistinct print especially if the original pages were typed with a poor typewriter ribbon or if the university sent us an inferior photocopy.

Reproduction in full or in part of this microform is governed by the Canadian Copyright Act, R.S.C. 1970, c. C-30, and subsequent amendments.

AVIS

La qualité de cette microforme dépend grandement de la qualité de la thèse soumise au microfilmage. Nous avons tout fait pour assurer une qualité supérieure de reproduction.

S'il manque des pages, veuillez communiquer avec l'université qui a conféré le grade.

La qualité d'impression de certaines pages peut laisser à désirer, surtout si les pages originales ont été dactylographiées à l'aide d'un ruban usé ou si l'université nous a fait parvenir une photocopie de qualité inférieure.

La reproduction, même partielle, de cette microforme est soumise à la Loi canadienne sur le droit d'auteur, SRC 1970, c. C-30, et ses amendements subséquents.

Canada

The Aerodynamics of Turbine Blades with Trailing Edge Damage

by

David Paul Isaacs

B. Eng. (Mechanical Engineering)

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

Master of Engineering
in Mechanical Engineering

**Department of Mechanical and Aerospace Engineering
Ottawa–Carleton Institute
for Mechanical and Aerospace Engineering
Carleton University
Ottawa, Ontario
January 1994**

©Copyright

1994, David P. Isaacs

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 ACCORDE UNE LICENCE
IRREVOCABLE ET NON EXCLUSIVE
PERMETTANT A LA BIBLIOTHEQUE
NATIONALE DU CANADA DE
REPRODUIRE, PRETER, DISTRIBUER
OU VENDRE DES COPIES DE SA
THESE DE QUELQUE MANIERE ET
SOUS QUELQUE FORME QUE CE SOIT
POUR METTRE DES EXEMPLAIRES DE
CETTE THESE A LA DISPOSITION DES
PERSONNE INTERESSEES.

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 PROPRIETE
DU DROIT D'AUTEUR QUI PROTEGE
SA THESE. NI LA THESE NI DES
EXTRAITS SUBSTANTIELS DE CELLE-
CI NE DOIVENT ETRE IMPRIMES OU
AUTREMENT REPRODUITS SANS SON
AUTORISATION.

ISBN 0-315-98580-1

Name The Aerodynamics of Air Turbine Blades with Trailing Edge Damage
 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 0722
 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
 Special 0340
 Sociology of 0529
 Teacher Training 0530
 Technology 0710
 Tests and Measurements 0288
 Vocational 0747

LANGUAGE, LITERATURE AND LINGUISTICS

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

PHILOSOPHY, RELIGION AND THEOLOGY

Philosophy 0422
 Religion 0318
 General 0321
 Biblical Studies 0319
 Clergy 0320
 History of 0322
 Philosophy of 0469
 Theology 0323

SOCIAL SCIENCES

American Studies 0324
 Anthropology 0326
 Archaeology 0327
 Cultural 0310
 Physical 0272
 Business Administration 0770
 General 0454
 Accounting 0338
 Banking 0385
 Management 0501
 Marketing 0503
 Canadian Studies 0505
 Economics 0508
 General 0509
 Agricultural 0510
 Commerce-Business 0511
 Finance 0358
 History 0366
 Labor 0351
 Theory 0578
 Folklore 0366
 Geography 0351
 Gerontology 0578
 History 0578
 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 0615
 General 0616
 International Law and Relations 0617
 Public Administration 0814
 Recreation 0452
 Social Work 0626
 Sociology 0627
 General 0938
 Criminology and Penology 0631
 Demography 0628
 Ethnic and Racial Studies 0629
 Individual and Family Studies 0630
 Industrial and Labor Relations 0700
 Public and Social Welfare 0344
 Social Structure and Development 0709
 Theory and Methods 0999
 Transportation 0453
 Urban and Regional Planning 0453
 Women's Studies 0453

THE SCIENCES AND ENGINEERING

BIOLOGICAL SCIENCES

Agriculture 0473
 General 0285
 Agronomy 0475
 Animal Culture and Nutrition 0476
 Animal Pathology 0359
 Food Science and Technology 0478
 Forestry and Wildlife 0479
 Plant Culture 0480
 Plant Pathology 0817
 Plant Physiology 0777
 Range Management 0746
 Wildlife Technology 0306
 Biology 0287
 General 0308
 Anatomy 0309
 Biostatistics 0379
 Botany 0329
 Cell 0353
 Ecology 0369
 Entomology 0793
 Genetics 0410
 Limnology 0307
 Microbiology 0317
 Molecular 0416
 Neuroscience 0433
 Oceanography 0821
 Physiology 0778
 Radiation 0472
 Veterinary Science 0786
 Zoology 0760
 Biophysics 0786
 General 0760
 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
 Palynology 0427
 Physical Geography 0368
 Physical Oceanography 0415

HEALTH AND ENVIRONMENTAL SCIENCES

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

Speech Pathology 0460
 Toxicology 0383
 Home Economics 0386

PHYSICAL SCIENCES

Pure Sciences

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

Applied Sciences

Applied Mechanics 0346
 Computer Science 0984

Engineering 0537
 General 0538
 Aerospace 0539
 Agricultural 0540
 Automotive 0541
 Biomedical 0542
 Chemical 0543
 Civil 0544
 Electronics and Electrical 0348
 Heat and Thermodynamics 0545
 Hydraulic 0546
 Industrial 0547
 Marine 0794
 Materials Science 0548
 Mechanical 0743
 Metallurgy 0551
 Mining 0552
 Nuclear 0549
 Packaging 0765
 Petroleum 0554
 Sanitary and Municipal 0790
 System Science 0428
 Geotechnology 0796
 Operations Research 0795
 Plastics Technology 0994
 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 under signed recommend to the Faculty of Graduate Studies
and Research acceptance of the thesis

The Aerodynamics of Turbine Blades with Trailing Edge Damage

submitted by David Paul Isaacs
in partial fulfillment of the requirements for
the degree of Master of Engineering
in Mechanical Engineering

Thesis Supervisor S.A. Sjolander

Chair, Department of Mechanical and Aerospace Engineering

Carleton University

January 1994

Abstract

In-service damage to compressor or turbine blades is a common cause of engine deterioration. Such damage can take a number of forms, such as: surface roughening; tip damage; leading edge damage; or trailing edge damage. Extending the capabilities of existing gas turbine Engine Health Monitoring (EHM) systems, in order to distinguish these forms of damage from one another, requires knowledge of the aerothermodynamic effects which might be specific to each damage mode.

The present research investigated the aerodynamic effects of trailing edge damage on a large-scale linear cascade of turbine blades, concentrating on the blades' loss characteristics and flow turning ability. Trailing edge damage was modelled by a semi-circular cutout located at the midspan of the trailing edge. Using a low-speed cascade wind tunnel, the effects of cutout diameters of 15% and 25% of the blade chord have been investigated. Measurements were made at incidences of -10° , 0° (design), $+5^\circ$, and $+10^\circ$. In addition, the experimental cascade was modelled using a computational Navier-Stokes solver developed by Dawes (1986), mainly to test the ability of the Dawes' solver to predict the flow through a cascade of blades with trailing edge damage. Solutions for a 25% cutout at incidences of -10° , 0° (design), and $+10^\circ$ have been calculated, with flow conditions ranging from high subsonic to transonic.

The experimental investigation found that the effect of trailing edge damage on the turbine cascade was subtle. The flow turning ability of the blades was found to be more affected than the total pressure losses. The computational results were encouraging. Although there were some obvious discrepancies, the Dawes code captured the essential features of the flows, suggesting that the code has the potential to become a powerful tool for investigating the aerodynamics of damaged turbine blades.

Acknowledgments

I would like to thank Professor S.A Sjolander for his invaluable support through all the phases of my Master's research. The input of his extensive experience put me back on the correct path many times throughout my experimental and computational endeavours.

My experimental work could not have been completed without the assistance of Mike Abbott, who showed me how to use the wind tunnel, and put up (and even assisted) with my constant messing around with the data acquisition system. He also showed extreme patience when experiments went longer than predicted. I must also thank Dekun Cao, Nathalie Hetu, Wendy Klein, and David Whitehouse for their assistance with my understanding of the data acquisition system and data reduction procedures.

I would like to thank Ian Jefferson for showing me Unix is better, C is good, and why I should buy NeXTStep, and also for putting up with my endless queries and requests. Also on the computer side, I would like to thank Rick Mallett from CCS, for providing much assistance with the resources of the network, and for actually helping provide more disk space.

The assistance I received throughout my research from Fred Barrett, Bruce Johnston, the staff of the Department of M&A Engineering Office, and the technicians of ETC Mechanical was greatly appreciated.

The financial support of the Natural Science and Engineering Research Council, GasTOPS Limited, and the Government of Ontario under the University Research Incentive Fund. is gratefully acknowledged.

I would like to thank my parents for all their encouragement, love, and support through all my academic endeavours. I could not have done so much without them.

Finally, I would like to thank Ruby, my beloved wife and best friend, for being there with her constant love and support during my academic work. I Love You Very Much.

Contents

1	Introduction	1
2	Background	3
2.1	Common In-Service Damage in Axial Flow Turbines	3
2.2	Engine Health Monitoring	4
2.2.1	Definition of Engine Health Monitoring	4
2.2.2	Engine Performance Simulation	5
2.3	Turbine Cascade Terminology	13
2.4	Computational Fluid Dynamics	13
2.4.1	The Dawes Code	15
2.5	Review of Previous Studies	19
3	Experimental Apparatus and Procedure	25
3.1	Cascade Wind Tunnel Apparatus	25
3.1.1	Cascade Wind Tunnel	25
3.1.2	Cascade Test Section	25
3.1.3	Undamaged and Damaged Blade Geometry	28
3.2	Instrumentation and Data Acquisition	28
3.2.1	Three-Hole Pressure Probe	30
3.2.2	Seven-Hole Pressure Probe	32
3.2.3	Two-Transducer Data Acquisition Setup	33

3.2.4	Four-Transducer Data Acquisition Setup	33
3.2.5	Probe Mounting and Traversing	37
3.3	Data Acquisition and Reduction	40
3.3.1	Data Acquisition Software	40
3.3.2	Data Reduction Software	41
3.4	Establishing Baseline Cascade Performance	43
3.4.1	Inlet Flow Angle	43
3.4.2	Outlet Flow Periodicity	43
3.4.3	Axial Velocity Ratio	47
3.4.4	Width of Uniform Low Loss Region	47
3.5	Measuring Damaged Blade Performance	49
3.5.1	Establishing Spanwise Symmetry of Downstream Flow	49
3.5.2	Establishing Magnitude of Downstream Flow Angularity . . .	51
4	Computational Analysis	54
4.1	Establishing Behaviour of the Dawes Code for an Undamaged Cascade	55
4.1.1	Number of Time-Steps to Steady-State Conditions	55
4.1.2	Uniform Flow Region	58
4.1.3	Effects of Grid Density on Undamaged Cascade	60
4.1.4	Effects of Increased and Decreased Cascade Mach Number . .	62
4.2	Development of Grid Modifications to Simulate Trailing Edge Damage	67
4.2.1	Generating a Damaged Grid Description	67
4.2.2	Modifications of Trailing Edge Geometry	69
4.3	Pre- and Post-Processing Software	75
4.3.1	The CASTLE Preprocessor	75
4.3.2	Post-Processors: Extract and Centrepont	77
5	Experimental and Computational Results	79
5.1	Experimental Results	79

5.1.1	Measured Wake Profiles	79
5.1.2	Spanwise Distribution of Pressure Loss and Deviation	81
5.1.3	Mass-Averaged Results	84
5.2	Computational Results	86
5.2.1	Overview of Calculated Flow Field	86
5.2.2	Spanwise Distribution of Pressure Loss and Deviation	89
5.2.3	Mass-Averaged Results	89
5.2.4	Effects of Increased Mach Number	92
6	Conclusions and Recommendations	100
A	Additional Experimental Apparatus and Procedure Information	106
A.1	Data Acquisition Equipment List	107
A.2	Windtunnel Checklist	108
B	The Castle Preprocessor	110
B.1	Overview of the Castle Program	111
B.1.1	Starting a Castle Session	111
B.1.2	Interacting with Castle	111
B.1.3	The Set, Show, and Plot Commands	112
B.2	The Major Grid Sections and Directions	113
B.3	Grid Clustering	116
B.3.1	Clustering in the Pitchwise and Spanwise Directions	116
B.3.2	Streamwise Clustering	119
B.4	Adding Blade Damage	120
B.4.1	Setting the Damage Size and Location	121
B.4.2	Spanwise Grid Clustering and Blade Damage	123
B.5	The Profile Coordinates File	124
B.6	The castle.defaults File	125
B.7	An Example Castle Session	126

B.8	Glossary of all Castle Parameters	129
B.8.1	Geometric Parameters	129
B.8.2	Boundary, Flow Condition, and Numerical Parameters	132
B.9	Sample File Listings	135
C	Description of The Dawes Code Post-Processor, Centrepoint	140
C.1	Overview of the Centrepoint Program	141
C.1.1	Starting a Centrepoint Session	141
C.1.2	Interacting with Centrepoint	141
C.2	The Executive Menu	142
C.3	The Extract Flow Data Menu	144
C.4	Centrepoint Output Files	152

List of Figures

2.1	First Stage Turbine Nozzle with Trailing Edge Damage	4
2.2	Overview of Engine Health Monitoring Data Processing	6
2.3	Turbine Performance Characteristics	7
2.4	Altering Turbine Efficiency to Model Turbine Faults	8
2.5	Breakdown of Engine Simulation, Focussing on Turbine Performance Simulation	12
2.6	Conventional Turbine Cascade Terminology	14
2.7	H-mesh Used by the Dawes Code	17
2.8	Compressor Cascade Blades with Crenulated Trailing Edges	20
2.9	Flow Turning Angle for Crenulated and Uncrenulated Blades	21
2.10	Downstream Pressure Loss Ratio for “Crenulated” Blades	22
2.11	Spanwise Distribution of Total Pressure Loss Measured by Klein (1993)	23
2.12	Spanwise Distribution of Outlet Flow Angle Measured by Klein (1993)	23
2.13	Counter-rotating Vortices Shed from Trailing Edge Cutout (Klein 1993)	24
3.1	Schematic of Wind Tunnel	26
3.2	Variable Incidence Test Section	27
3.3	Summary of Cascade Blade Geometry	29
3.4	Three-Hole Probe Geometry	31
3.5	Seven-Hole Probe Geometry	32
3.6	Two-Transducer Data Acquisition Setup	34

3.7	Transients in Three-Hole Probe Measurements After Scanivalve Step	35
3.8	Four-Transducer Data Acquisition Setup	36
3.9	Inlet and Outlet Probe Mounting.	38
3.10	Inlet and Outlet Traverse Planes.	39
3.11	Acceptable Inlet Flow Angle Measurement	44
3.12	Acceptable Pressure Loss Periodicity Measurements	45
3.13	Acceptable Flow Angle Periodicity Measurements	46
3.14	Spanwise Loss Distribution Showing Uniform Loss Region	48
3.15	Spanwise Pressure Loss Distribution	50
3.16	Pitch Angle Measured Downstream of Damaged Blade	52
3.17	Yaw Angle Measured Downstream of Damaged Blade	53
4.1	Residue vs. Time-Steps	56
4.2	Pitchwise Mass-Averaged Total Pressure Loss Distribution at Various Time-Steps	57
4.3	Pitchwise Mass-Averaged Total Pressure Loss Distribution, Showing Excluded Endwall Region	59
4.4	Pitchwise Mass-Averaged Total Pressure Loss Distribution for Various Grid Densities, after 1250 Time-Steps	61
4.5	Leading and Trailing Edge Mach Number vs. Inlet Total Pressure . .	62
4.6	Contours of Constant Mach Number	64
4.7	Contours of Constant Mach Number	65
4.8	Mass-Averaged Total Pressure Loss vs. Trailing Edge Mach Number .	66
4.9	Schematic of the Blade Profile Description Used by the Dawes Code .	68
4.10	Damaged Blade Profile Description.	70
4.11	Finite Volume Cells Near Damaged Trailing Edge, Viewed from Stream- wise Surface at Midspan	71
4.12	Finite Volume Cells Near Damaged Trailing Edge, Viewed from Merid- ional Surface	72

4.13	Rounding of the Top and Bottom Edges of Cutout.	73
4.14	Mass-Averaged Total Pressure Loss Distribution for Various Numbers of Streamwise Surfaces Across Cutout	74
4.15	Computational Grid ($33 \times 107 \times 33$) for Trailing Edge Damaged Blade	76
5.1	Total Pressure Losses Measured in Damaged Blade Wakes	80
5.2	Pitchwise Mass-Averaged Total Pressure Loss for Various Incidence Angles.	82
5.3	Pitchwise Mass-Averaged Trailing Edge Deviation for Various Inci- dence Angles.	84
5.4	Mass-Averaged Total Pressure Loss for Blade with Unit Aspect Ratio.	85
5.5	Contours of Constant Mach Number at Midspan	87
5.6	Contours of Constant Mach Number at $Z/R_d \approx 4$	88
5.7	Pitchwise Mass-Averaged Total Pressure Loss for Various Incidence Angles: Dawes Code and Measured Results.	89
5.8	Pitchwise Mass-Averaged Trailing Edge Deviation for Various Inci- dence Angles: Dawes Code and Measured Results.	90
5.9	Dawes Code Predictions of Mass-Averaged Total Pressure Loss for a Blade with Unit Aspect Ratio.	92
5.10	Contours of Constant Mach Number at Midspan	94
5.11	Contours of Constant Mach Number at $Z/R_d \approx 4$	95
5.12	Pitchwise Mass-Averaged Total Pressure Loss for Two Transonic Mach Numbers	96
5.13	Pitchwise Mass-Averaged Trailing Edge Deviation for Two Transonic Mach Numbers	97
5.14	Mass-Averaged Total Pressure Loss for Damaged and Undamaged Blade	98
A.1	Cascade Wind Tunnel Checklist	108

B.1	The Grid Terminology used by Castle: (top) streamwise mesh;(bottom) meridional mesh.	115
B.2	Castle Clustering Types: (a) one-sided clustering; (b) two-sided clustering; (c) no clustering.	117
B.3	Damage Size and Location Parameters	122
B.4	The Three Damaged Grid Sections (indicated on meridional mesh with blade edges highlighted)	123
B.5	Specifying Blade Profile Coordinates	125
C.1	Example of GNUPLOT contour plot of Mach number.	155

List of Tables

2.1	Effects of Tip Clearance on Turbine Performance Calculated Using Streamline Curvature Method	10
4.1	Summary of Grid Density Runs.	60
A.1	Data Acquisition Equipment	107

List of Symbols

AVR	axial velocity ratio
C	blade chord length
C_p	pressure coefficient
C_x	blade axial chord length
E	total fluid energy
H	blade span
i	cascade incidence angle ($i = \alpha_1 - \alpha_{1des}$)
I, J, K	grid indices for the bladewise, streamwise, and spanwise directions, respectively
$I_{max} \times J_{max} \times K_{max}$	grid dimensions in bladewise, streamwise, and spanwise directions, respectively
$\dot{m}$	mass flow rate
M	Mach number
P	pressure
q	dynamic pressure
S	blade spacing
r	radial coordinate
$r\theta$	circumferential coordinate (sector length)
R	radius
u, v, w	velocity components
V	velocity ($V^2 = u^2 + v^2 + w^2$)
x	axial coordinate
Y	1. cascade bladewise coordinate 2. total pressure loss coefficient

Y	mass averaged total pressure loss coefficient
Z	cascade spanwise coordinate
α	pitch angle (relative to pressure probe)
α_1	inlet flow angle
α_2	outlet flow angle
β	yaw angle (relative to pressure probe)
β_1	inlet metal angle
β_2	outlet metal angle
δ	trailing edge deviation ($\delta = \alpha_2 - \beta_2$)
γ	stagger angle
θ	1. meta turning angle ($\theta = \beta_1 - \beta_2$) 2. cylindrical coordinate

Additional Subscripts

a	atmosphere
$axial$	axial direction
c	corner
d	damage
des	design
le	blade leading edge
in	upstream of cascade
o	total (stagnation) conditions
s	static conditions
te	blade trailing edge
up	upstream of cascade
$scani$	from Scanivalve
x	axial direction

Superscripts

' pitchwise mass averaged value

Chapter 1

Introduction

Essentially all civilian and military users of gas turbine engines routinely use engine health monitoring (EHM) systems (e.g. Muir et al 1988, Cue et al 1991). EHM systems alert engine operators to gradual or abrupt engine deterioration, so that rectifying action can be taken to avoid severe engine damage or unexpected shutdown of the engine. Therefore EHM systems allow safer, more economical operation of the engine. Unfortunately, existing EHM systems are only able to identify the presence of a problem and the general area of the engine where the problem exists, but not the specific nature of the problem. As a result, diagnosing the specific nature of the problem becomes a significant fraction of the engine maintenance time.

In-service damage to compressor or turbine blades is a common cause of engine deterioration. Such damage can take a number of forms: surface roughening due to fouling, erosion, or corrosion; tip, leading edge, and trailing edge damage due to foreign object ingestion, over-temperature, excessive vibration, or blade cooling problems; and a few other types of problems. In order to distinguish these types of damage from one another, a knowledge of the aerothermodynamic effects which might be specific to each damage mode is required. At present, very few of these aerothermodynamics effects are known. Therefore, it is necessary to begin at the blade level, with an understanding of the effect of a particular mode of damage on the loss characteristics and flow turning ability of a blade. From such information,

the stage and ultimately the engine performance, incorporating the effects of the damage, can be predicted. An outline of EHM and how research data is incorporated into engine performance simulation in support of EHM is included in Chapter 2. This chapter also includes a review of previous studies related to the present research.

The objective of the present research was to provide data concerning the loss characteristics and flow turning ability of turbine blades with trailing edge damage. This data is to be used to assist the development of correlations describing the effects of damage on turbine blade aerodynamics. The present research investigated the aerodynamic effects of trailing edge damage on a large scale linear cascade of turbine blades. Trailing edge damage has been modelled by a semi-circular cutout located at the midspan of the trailing edge. Using a low-speed cascade wind tunnel, the effects of cutout diameters of 15% and 25% of the blade chord have been investigated. Measurements were made at design incidence, and at off-design incidences of -10° , $+5^\circ$, and $+10^\circ$. Details concerning the experimental apparatus and procedures used in the present research are contained in Chapter 3.

Investigations were also made using a computational 3-Dimensional Navier-Stokes solver developed by Dawes (1986b). Using Dawes' code, a cascade geometrically similar to the experimental cascade has been modelled. Solutions for a 25% cutout at incidences of -10° , 0° (design), and $+10^\circ$ have been calculated, with flow conditions ranging from high subsonic to transonic. The primary objective of the Dawes code calculations was not to provide definitive data, but to assess the ability of the Dawes code to predict the aerodynamics of turbine blades with trailing edge damage. How the Dawes code was used in the present research is described in detail in Chapter 4. Both the experimental and computational results are presented, with accompanying discussion, in Chapter 5. The overall conclusions and recommendations for further study are summarized in Chapter 6.

Chapter 2

Background

2.1 Common In-Service Damage in Axial Flow Turbines

In-service damage in gas turbine engines can take a number of forms and be due to a number of causes. Excessive erosion or corrosion can result in a roughening of the blade surfaces. Ingestion of a foreign object can be very destructive, causing damage to blade leading edges, tips, and trailing edges. Vibration problems can result in wear of the blade tips, causing an increase in tip clearance. Over-temperatures and blade cooling problems can melt blade leading and trailing edges. The common modes of in-service damage are best grouped into four categories: surface roughening; tip damage; leading edge damage; and trailing edge damage. Each category is best studied separately to establish its specific impact on the blade aerodynamics.

In the present research, trailing edge damage will be modelled by the presence of a semi-circular cutout at midspan on the trailing edge of a cascade blade, as described in detail in Chapter 3. This can be considered a very idealistic representation of trailing edge damage, given an actual example shown in Figure 2.1.

Figure 2.1: First Stage Turbine Nozzle with Trailing Edge Damage. (From Mattheij 1985)

2.2 Engine Health Monitoring

The ongoing goal of Engine Health Monitoring (EHM) is the improvement of the reliability of an engine, increasing the vehicle safety and reducing the engine maintenance costs. Turyk (1989) gives an excellent review of engine health monitoring, from which much of the following discussion is derived.

2.2.1 Definition of Engine Health Monitoring

Applied to gas turbine engines, EHM deals with the thermodynamic as well as the mechanical aspects of engine operation. This includes monitoring overall engine performance, and combining gas-path, oil system, and vibration analysis to determine the overall "health" of an engine. Recognizing that engine maintenance is expensive, compounded by the expense of the engine downtime itself, the ultimate goal of EHM is to be able to predict not only when engine performance is suffering, but to what degree and why.

To determine why a gas turbine engine's performance is suffering requires the use

of an extensive *fault library of fault signatures* which characterize the effect of *faults* to the engine. Many faults encountered in gas turbine engines are due to in-service damage, as described in Section 2.1. A fault signature is established by comparing the performance of an engine with a known fault with the baseline engine performance (i.e. that of the healthy engine). By comparing the performance signature of an in-service engine with the fault library, performance degradation can be isolated and appropriate maintenance action recommended. An overview of the data processing is shown schematically in Figure 2.2.

2.2.2 Engine Performance Simulation

For the most part, determining the fault signature associated with a specific fault requires detailed and accurate numerical simulation of the engine performance. It is impractical, and often impossible, to test experimentally for the effects of many types of faults on a physical engine, due to factors such as expense, safety, and time constraints. Data can also be derived from maintenance data bases, which catalogue the fault signatures of in-service engines with well diagnosed faults, but large gaps must still be filled using numerical simulation.

To simulate the overall performance of a gas turbine engine, mathematical models which calculate the performance of significant components (e.g. compressor, combustor, turbine) are required. The performance of each component is calculated separately, and a central component matching algorithm links the components together to derive the overall engine performance.

Most turbine component models are simply a description of the turbine characteristics, that is representations of turbine efficiency and pressure ratio as some function of rotor speed and mass flow. Figure 2.3 shows a commonly used form of turbine characteristics, in graphical form. For example, from this figure, given the mass flow coefficient, $m\sqrt{T_{03}}/P_{03}$, and the corrected rotor speed, $N/\sqrt{T_{03}}$, one could find the corresponding turbine pressure ratio and efficiency. Most models

Figure 2.2: Overview of Engine Health Monitoring Data Processing (After Muir et al 1988)

Figure 2.3: Turbine Performance Characteristics (After Cohen et al 1987 *Gas Turbine Theory*)

implemented by computer would look up the required values in a table version of the turbine characteristics. By altering the turbine characteristics the effects of different faults on the turbine, and ultimately the engine performance, can be determined. Figure 2.4 shows a couple of examples of how turbine efficiency might be altered to model the effects of a turbine fault (or faults). However, for most blade damage type faults, the links between specific faults and the appropriate alterations to the turbine characteristics have not yet been established.

Mean-line analysis is one approach which may be useful for incorporating the effects of blade damage in turbine performance predictions. Existing correlations, such as Ainley and Mathieson (1957) and Kacker and Okapuu (1981), are used to establish the flow losses such as profile losses, shock losses, secondary flow losses, and tip clearance losses through each blade passage. These correlations require as

Figure 2.4: Altering Turbine Efficiency to Model Turbine Faults

input the basic midspan blade geometry (e.g. solidity, maximum thickness to chord ratio, aspect ratio, leading/trailing edge radii) of each blade row and the *relative* flow information (e.g. inlet/outlet flow angles, Mach number, Reynolds Number). The flow information is obtained by analysis of the midspan velocity triangles at each stage. Starting at the first stage, the correlations are used to calculate the flow losses through the stage (nozzle and rotor) from which the stage efficiency and pressure and temperature drop are calculated. The flow conditions at the outlet of the first stage become the inlet condition for the second stage, and so on.

Mean-line analysis has been used in turbine design and analysis for many years, and is thus a well established and proven tool. However, for the purposes of predicting the performance of turbines with blade damage, it does have some disadvantages. The primary disadvantage is that losses are inherently averaged over the entire span of a blade row. This implies that there is no way of distinguishing the effects of one mode of loss from another. For example, if the added losses due to trailing edge damage happen to be the same magnitude as the added losses due to tip damage, mean-line analysis will predict the same turbine performance, leading ultimately to

the same engine fault signature for both cases. It may be possible to circumvent this problem by comparing off design losses as well as design losses. Losses that may be equal at design incidence may not, in general, be equal at off-design incidences. By considering both design and off design performance, it may be possible to establish fault signatures unique to a mode of damage. This would allow different modes of damage to be distinguished from one another, at the price of monitoring two operating points instead of one.

A more involved approach to predicting turbine performance is the streamline curvature method. This approach is two-dimensional, calculating in the radial and streamwise direction (the mean-line analysis can be thought as one-dimensional, calculating in the streamwise direction only). Although the bulk of previous studies utilizing the streamline curvature method have concentrated on compressor analysis, studies concentrating on turbine analysis do exist (e.g Ngo and Millar (1973), Klein (1993)). The governing equations used are the radial equilibrium equation, continuity and energy equations, and the equation of state. Using a numerical, iterative algorithm, the trajectories and locations of the flow streamlines are solved, with the number of streamlines to position as one of the user inputs. Pressure and temperature drops are derived utilizing the same loss correlations used with the mean-line analysis.

In the past, streamline curvature methods applied to turbines have distributed all losses evenly across the span of the blades. Klein (1993) modified this approach by concentrating the losses due to tip clearance near the tip of the blade. The tip leakage loss was distributed in a parabolic form, following tip leakage measurements made by Yaras (1987). Table 2.1 summarizes Klein's results for calculations using concentrated and distributed tip clearance loss, for two different tip clearances. The concentrated loss calculations show a slightly less, but still significant, change in performance with increasing tip clearance when compared with the distributed loss calculations. This is encouraging from the EHM perspective, since significant changes may be detectable by existing engine monitoring systems. Klein also observed that,

Table 2.1: Effects of Tip Clearance on Turbine Performance Calculated Using Streamline Curvature Method (after Klein 1993).

Comparison of Turbine Performance With and Without Concentrated Tip Loss For a High Temperature, Pressure Turbine				
Inlet Temperature	1627.8 K			
Inlet Pressure	337 psi			
Mass Flow	92.3 lb _m /sec			
Speed	16200 rpm			
	Distributed Tip Loss TCL = 0.03"	Concentrated Tip Loss TCL = 0.03"	Distributed Tip Loss TCL = 0.05"	Concentrated Tip Loss TCL = 0.05"
Total Pressure Ratio	2.399	2.372	2.728	2.537
Total Temperature Ratio	1.191	1.188	1.217	1.201
Total Temperature Drop	261 K	258 K	290 K	272 K
Total-Total Efficiency	86.39%	86.44%	84.93%	85.29%
Power Output	17929 HP	17725 HP	19919.8 HP	18710.4 HP

for the concentrated loss calculations, the temperature drop through the turbine was less near the tip than it was near the hub. For the distributed loss calculations the temperature drop was approximately constant across the blade span. This change in the temperature distribution suggests a method of distinguishing types of damage from one another. By measuring temperatures with strategically located thermocouples in an actual turbine, local temperature variations specific to a type of damage may be detected, leading to an accurate diagnosis of the turbine damage. Klein concluded that the streamline curvature method possessed excellent potential for predicting turbine performance for EHM purposes.

None of the correlations available for use with the mean-line analysis and the streamline curvature method incorporate the effects of damage to turbine blades. Creating such correlations will require a detailed knowledge of the aerothermodynamic effects of damage to turbine blades. This is the goal of the current research project initiated at Carleton University, of which the research described in this thesis is a part. Figure 2.5 expands on the Engine Simulation node of Figure 2.2, to illustrate how research data will be incorporated into correlations and used to predict turbine performance. The question marks (?) indicate the fact that none of the turbine performance prediction methods have yet been established.

As illustrated in Figure 2.5, advanced calculations methods, such as multistage Navier-Stokes calculations, will not require loss correlations but be able to predict turbine performance directly. Such calculation methods represent the forefront of CFD research, and are very complex and computationally intensive. It is doubtful that such an approach will become practical in the near future, given the simpler and much less computationally demanding mean-line analysis and streamline curvature methods. A more modest use of three-dimensional Navier-Stokes solvers is the prediction of the flow through a single row of blades, such as a linear cascade. Data gained from such solutions can be used in the same way physical cascade data is used: to help establish aerodynamic correlations for use with mean-line analysis or streamline

Figure 2.5: Breakdown of Engine Simulation, Focussing on Turbine Performance Simulation

curvature methods.

2.3 Turbine Cascade Terminology

The terminology conventionally used in describing an axial turbine cascade is outlined in Figure 2.6. The *design inlet flow angle*, α_{1des} is chosen by the blade designer as the angle at which aerodynamic losses are at a minimum. This is generally not equal to the inlet metal angle, and the *design incidence*, i_{des} , is defined as the angle between the design inlet flow angle and the inlet metal angle (i.e. $i_{des} = \alpha_{1des} - \beta_1$). For example, the design incidence of the cascade used for the present research is $i_{des} = 3^\circ$. Figure 2.6 defines the general incidence as the difference between the inlet flow angle and the inlet metal angle. The incidence can also be referenced to the design inlet flow angle, in which case the incidence is defined as the angle between the inlet flow angle and the design inlet flow angle (i.e. $i = \alpha_1 - \alpha_{1des}$). The latter definition was used in the present research.

The cascade used in the present research is described in detail in Chapter 3.

2.4 Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) can be defined very broadly as the prediction of fluid flow characteristics by the numerical solution of the governing equations of fluid dynamics. The prediction of turbomachinery flows is especially challenging, given the compressible, highly turbulent, and three-dimensional nature of the flows. Nevertheless, considerable progress has been made in the development of three-dimensional Navier-Stokes solvers. One such code, described in the next section, has been used as a tool in the present research.

Figure 2.6: Conventional Turbine Cascade Terminology

2.4.1 The Dawes Code

The Navier-Stokes solver used in the present research was developed by Dawes (e.g. Dawes 1986a,b) at the Whittle Laboratory, Cambridge University, England, and is specifically intended for turbomachinery (both axial and radial) calculations. The Dawes code solves the three-dimensional continuity equation, Reynolds averaged Navier-Stokes equations, and energy equation cast in a finite volume form in the blade-relative frame of reference, using a cylindrical (r, θ, x) coordinate system:

$$\frac{\partial}{\partial t} \oint_{Vol} \bar{U} dVol = \oint \bar{H} d\overline{Area} + \rho \bar{S} \quad (2.1)$$

where

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

with

$\bar{q} = V_x \hat{i}_x + V_r \hat{i}_r + V_\theta \hat{i}_\theta$, the relative velocity

Ω , the rotational speed

$\bar{\tau}$, the stress tensor containing both the static and the viscous stresses.

$I = c_p T_{o,rel} - \frac{1}{2}(\Omega r)^2$, the rothalpy

The system is closed by an equation of state $p = \rho(\gamma - 1)(E - 0.5 * (\bar{q} \cdot \bar{q} - (\Omega r)^2)$. The Dawes code has the following features: a time marched steady-state solution of the Reynolds averaged Navier-Stokes equations (above); a structured H-mesh; the Baldwin-Lomax mixing length turbulence model; one level of multigrid acceleration of the time marched solution; and adaptive artificial viscosity to enable stable capture of shock waves. These features are described in more detail in the following sections.

Time Marched Steady-State Solution

The Dawes code attempts to achieve an asymptotic time-steady solution of its governing equations using an implicit time marching algorithm. Starting from an initial guess based on the input boundary conditions, the solution of the governing equations (Equation 2.1) is calculated moving in discrete time-steps (hence the term “time marching”) until a steady solution is achieved. Since the intermediate solutions have no physical relevance, each finite volume cell of the mesh is calculated using its own time step based on the local CFL (Courant-Friedrichs-Lewy) limit. This reduces the number of time steps necessary to achieve steady-state, without compromising the integrity of the steady-state solution.

Computational Mesh

The Dawes code utilizes an H-mesh, chosen by Dawes for its simplicity and flexibility. Such a mesh is formed by the intersection of three families of surfaces: streamwise, quasi-orthogonal, and meridional. The streamwise surfaces are surfaces of revolution, the first and last surfaces coinciding with the hub and casing respectively. The first and last meridional surfaces coincide with the blade suction and pressure surfaces respectively. Examples of streamwise and meridional surfaces are shown in Figure 2.7, which also shows the $I \times J \times K$ nomenclature used to identify individual finite volume cells. Note that Figure 2.7 shows the streamwise mesh doubled in the I direction so that the profile of the turbine blade can be seen. Also note that the H-mesh shown is a rather coarse and ill-designed mesh, prepared only as an example in which the finite volume cells are large enough to be seen clearly. A more practical mesh would have a much higher density. Also, Dawes (1986a) suggests that the inlet and outlet planes should be at least one-half a blade pitch (or one-half blade span, whichever is greater) up and downstream of the blade row, respectively, to allow proper development of the flow.

Figure 2.7: H-mesh Used by the Dawes Code: (top) Horizontal Projection of Streamwise Surface; (bottom) Vertical Projection of Meridional Surface.

Turbulence Model

Solutions to the Reynolds averaged Navier-Stokes equations require some sort of mathematical model of the Reynolds turbulent stress tensor in order to close the system of equations. The Dawes code implements a model patterned after Baldwin and Lomax (1978), a two layer model based on the Prandtl mixing length concept. This is a relatively low-order model, but Dawes (1986a) states that any model which predicts the correct amount of blockage in the correct areas of the blade passage is sufficient. There is no method of predicting the transition point on the blade. The Dawes code assumes that turbulent boundary layers exist on the blade surfaces right from the leading edge. When prescribing the wall shear stresses, the boundary layer is resolved down to the laminar sublayer scale and coupled with the known zero velocity on the wall to compute the velocity gradients. If the mesh is too coarse to resolve the boundary layer, a log-law skin friction correlation is invoked.

Multigrid Acceleration

The Dawes code allows for one level of multigrid to exist in order to accelerate the solution process. The multigrid procedure allows significant features of the overall flow pattern to propagate rapidly on a coarse grid, while fine details are resolved on a fine grid. Dawes found that the number of time steps to steady state can be halved by using one level of multigrid, with little improvement with the addition of a second level of multigrid (i.e. a fine grid, coarse grid, and extra-coarse grid). The Dawes code generates the coarse grid by deleting every other surface of the original H-mesh. The Dawes code also allows for the calculations to start using the coarse mesh only, for a specified number of time steps. This allows the quick generation of a relatively accurate first guess before starting the fine mesh calculations.

Artificial Viscosity

To ensure stability in regions with strong pressure gradients (e.g. through shock waves) an adaptive artificial viscosity parameter, patterned after Jameson (1984) is included in the discretized governing equations. The Dawes code provides three coefficients, which control of the artificial viscosity terms. They are applied to the solution in an adaptive manner: the artificial viscosity coefficients are only applied in areas where strong pressure gradients are detected (i.e. in shock waves). Dawes gives recommended values for the coefficients, and his values were used throughout the present research.

Chapter 4 describes in detail how the Dawes code was applied to the problem of the prediction of flow through a cascade of turbine blades with trailing edge damage.

2.5 Review of Previous Studies

Very little research has been performed to quantify the aerodynamic effects of damage to turbomachinery. Turyk (1989) includes a review of the status of research into the effects of damage to axial flow compressors. Three types of damage are considered: tip damage, surface roughness, and leading edge damage. The review concludes that for tip clearance, the amount of data available in the literature is extensive, and research is still very active. This conclusion can be extended to axial flow turbines, since a number of research groups, including the turbomachinery laboratory of the Department of Mechanical and Aerospace Engineering at Carleton University, have been very active in this area (e.g. Yaras, Sjolander, and Kind 1991). For surface roughness a fair amount of data exists, but more work needs to be done. The effects of leading edge deterioration was the least researched area. Most experimental research projects had involved sand ingestion tests on gas turbine engines, which leads to erosion of the blade leading edges, among other things. Turyk also recommended that research into the effects of trailing edge thinning be considered. The modes of

Figure 2.8: Compressor Cascade Blades with Crenulated Trailing Edges (From DeCook et al 1993)

leading edge and trailing edge damage considered by Turyk were generally of the erosive type: that is rounding or blunting of leading edges and thinning of trailing edges. These effects can be considered two-dimensional and are not really of the three-dimensional nature that a trailing edge cutout would have.

DeCook, King, and Elrod (1993) have researched the flow in a subsonic cascade of compressor blades with what the authors term “crenulated” trailing edges, illustrated in Figure 2.8. Their goal was to enhance downstream mixing of the wakes, which would reduce the flow unsteadiness for downstream blade rows and improve fuel/air mixing in the combustor. The crenulations were thought to enhance mixing by generating counter-rotating vortices, due to the pressure difference between the pressure and suction surfaces of the blades. Similar flow effects would be expected for both compressor and turbine blades where the trailing edge has become serrated due to damage, such as the nozzle blade shown in Figure 2.1. Figures 2.9 and 2.10 illustrate their typical results. The blade turning angle can be seen to decrease as the crenulation size increases. The authors concluded that the reduction in turning was most likely due to the missing blade curvature in the crenulations. The pressure loss ratio shown in Figure 2.10 represents the mass averaged pressure loss coefficient, $\bar{\omega} = (P_{o1} - P_{o2}) / (0.5 \rho_1 V_1^2)$, for the crenulated blades normalized by the mass averaged

Figure 2.9: Flow Turning Angle for Crenulated and Uncrenulated Blades (from DeCook et al 1993)

pressure loss coefficient, $\bar{w}_o$, for the unmodified blades. Interestingly, the authors observed decreased losses for the crenulated blades, but offered no explanation for this behavior.

The only measurements that seem to exist for turbine blades with damaged trailing edges were made recently by Klein (1993). The experimental portion of the present study is a continuation of Klein's work. Klein developed a method of casting high quality cascade blades from a mixture of epoxy and iron filings. These blades can be machined and thus modified to simulate trailing edge damage. Klein made preliminary measurements for a blade with a semi-circular cutout diameter 25% of chord, at the trailing edge midspan. Figure 2.11 and 2.12 show typical measurements of pitchwise mass-averaged pressure loss and outlet flow angle, measured 40% of an axial chord downstream. The pressure losses peak at the edges of the cutout, suggesting that two counter-rotating vortices are shed from the edges of the cutout, as shown in Figure 2.13.

It can be concluded that data on the effects of trailing edge damage in axial-flow turbomachinery is virtually nonexistent. The research project initiated at Carleton

Figure 2.10: Downstream Pressure Loss Ratio for "Crenulated" Blades (from DeCook et al 1993)

Figure 2.11: Spanwise Distribution of Total Pressure Loss Measured by Klein (1993)

Figure 2.12: Spanwise Distribution of Outlet Flow Angle Measured by Klein (1993)

Figure 2.13: Counter-rotating Vortices Shed from Trailing Edge Cutout (Klein 1993)

University represents the first look at these areas.

Chapter 3

Experimental Apparatus and Procedure

3.1 Cascade Wind Tunnel Apparatus

3.1.1 Cascade Wind Tunnel

A schematic of the wind tunnel used for this experiment is shown in Figure 3.1. The wind tunnel was commissioned by Rodger (1992). The air is drawn in through a bell-mouth inlet by a centrifugal fan driven by an AC motor. The motor speed is controlled by a remote variable frequency controller. Typically, the running speed of the fan is about 1500 RPM. The maximum exit velocity is about 30 m/s, which corresponds to a Mach number of about 0.09. Therefore, the flow through the cascade test section can be treated as essentially incompressible. The turbulence intensity at the test section inlet is about 0.3 percent. Periodic maintenance was performed on the tunnel following suggestions by Rodger (1992).

3.1.2 Cascade Test Section

The cascade test section used for this study is illustrated in Figure 3.2. The section was designed by Goobie (1989) to study the effect of several parameters such as the incidence, pitch to chord ratio, tip clearance, and aspect ratio on the aerodynamic performance of a rectilinear cascade of five blades. A brief description of the test

Figure 3.1: Schematic of Wind Tunnel (from Rodger 1992).

section is given below, for reference. For a more detailed description, see Goobie (1989).

The end-wall boundary layer thickness size is minimized by the presence of two adjustable boundary layer bleed slots located at the inlet of the test section. Endplates mounted on the blades allow for control of the axial velocity ratio by converging or diverging the flow through the blade passages as necessary. The endplates also reduce the intensity of secondary flows by allowing the end-wall boundary layers, which grow from the inlet bleed slots, to slide through the clearance between the endplates and the end-walls. There are seven additional control surfaces which allow adjustments to be made to the flow to meet requirements of flow uniformity at the inlet and flow periodicity at the outlet. A detailed description of how to use the control surfaces to adjust the cascade flow is given by Rodger (1992).

The blades themselves are mounted on a plate which is in turn mounted on the turntable. Therefore, cascades with different geometries can be installed relatively quickly by simply changing mounting plates. However, it should be noted that only a single cascade geometry was considered for the present experiment. The turntable is

Figure 3.2: Variable Incidence Test Section (from Sjolander et al 1993).

supported by a shaft mounted on two pillow block bearings, which allows the incidence to be varied by rotating the turntable. The range of available incidences is in excess of $\pm 45^\circ$. A protractor and vernier scale engraved on the back of the turntable enables the incidence to be adjusted with an accuracy of $\pm 0.1^\circ$.

The front wall of the test section is mounted on a separate frame and is removable to permit access to the blades and control surfaces. This wall is manufactured from plexiglass to allow good visual access to the flow, if necessary.

3.1.3 Undamaged and Damaged Blade Geometry

A summary of the blade and cascade geometry is shown in Figure 3.3. The cascade geometry is identical to that used in studies conducted by Whitehouse (1993), Klein (1993), and Abbott (1993). The blade profile represents a typical midspan section of a power turbine rotor blade of fairly recent design. The blades used in this study were cast by Klein from Devon Plastic Steel, a mixture of epoxy resin and iron filings. Six blades were cast. Five were left unmodified and the sixth had a semi-circular cutout with a diameter of 25% of the blade chord machined from the trailing edge, as shown in Figure 3.3. Later in the present research, one of the five undamaged blades had a similar cutout, with a diameter of 15% of the blade chord, cut in the trailing edge. An undamaged blade cast of the same material was obtained from Abbott to complete the undamaged cascade.

3.2 Instrumentation and Data Acquisition

The measurements taken during the present research were obtained using either three-holed or seven-holed pressure probes. A reference dynamic pressure was measured upstream of the blade row, by a pitot tube positioned outside of the boundary layer referenced to a static pressure tap located at the same distance upstream. All three-hole and seven-hole probe pressures were non-dimensionalized by this upstream

CHORD LENGTH, C	: 162.3 mm
BLADE SPAN, H	: 200 mm
BLADE SPACING, S	: 111 mm
MAXIMUM THICKNESS/C	: 0.196
STAGGER ANGLE, γ	: 21.6 deg.
INLET METAL ANGLE, β_1	: 25.5 deg.
OUTLET METAL ANGLE, β_2	: 57.5 deg.
LEADING EDGE DIAMETER	: 16.7 mm
TRAILING EDGE DIAMETER	: 4.2 mm

Figure 3.3: Summary of Cascade Blade Geometry (from Sjolander et al 1993).

dynamic pressure.

The following sections describe in detail the three-hole and seven-hole pressure probes, as well as the data acquisition setups that were used in performing the experiments. Information concerning equipment models and serial numbers is contained in Appendix A.

3.2.1 Three-Hole Pressure Probe

A drawing of the three-hole pressure probe used in the present research is shown in Figure 3.4. The three-hole probe measures flow velocity, yaw angle, and total pressure, and has been calibrated for yaw angles in the range of $\pm 10^\circ$. Tremblay (1989) and Goobie (1989) reported the accuracy of the measured yaw angle as within $\pm 0.5^\circ$ of the true yaw angle, and the accuracy of the measured dynamic pressure as within 1% of the local dynamic pressure. The calibration of the probe is considered valid for flows with pitch angles in the range of $\pm 10^\circ$. For this reason, at the start of the present research, the three-hole probe was used only to measure the wakes of the undamaged cascade. At the start of the present research, the magnitude of the flow angularity produced by a trailing edge damaged blade was not known. Only after measurements taken with the seven-hole probe had shown that the flow angularity was within $\pm 10^\circ$, for both yaw and pitch, was the three-hole probe used to measure the wakes of the damaged cascade (see Section 3.5). A shorter, but otherwise identical, three-hole probe was used to measure the inlet flow properties. The shorter probe was necessary to provide clearance of the back of the probe with the leading edge of the cascade blades.

Calibration of the three-hole probes was performed by Abbott (1993), once before the present research and once after the present research. It was found that the calibration of the three-hole probes remained essentially unchanged.

Note: scale 1:1

Figure 3.4: Three-Hole Probe Geometry (from Rodger 1992).

Figure 3.5: Seven-Hole Probe Geometry (from Yaras 1990).

3.2.2 Seven-Hole Pressure Probe

A diagram of the seven-hole pressure probe used in the experiments is shown in Figure 3.5. The seven-hole probe measures, with reasonable accuracy in flows that vary up to $\pm 50^\circ$ in both yaw and pitch, the flow velocity, total pressure, and both the yaw and pitch angles. The accuracy of the measured yaw and pitch angles is within $\pm 2^\circ$ of the true angles, while the accuracy of the measured pressures is within 8% of the local dynamic pressure.

In the present research the seven-hole probe was employed to ensure that the pitch angle downstream of the undamaged blades was small enough to allow accurate measurements using the three-hole probe (see Section 3.2.1). Overall, the seven-hole probe was not the instrument of choice, since the required experiment run time is 2

to 3 times longer using the seven-hole probe than using the three-hole probe.

The seven-hole probe requires a lengthy and involved calibration process. For this reason the seven-hole probe has been calibrated only once, by Danias (1987). However, the seven-hole probe has been often used for research over the years, and has had its calibration checked frequently. The seven-hole probe calibration was checked immediately prior to the present research. All errors were found to be within the tolerances specified above.

3.2.3 Two-Transducer Data Acquisition Setup

Initial testing on the wind tunnel was performed using a two-transducer setup to measure the pressures necessary for three-hole and seven-hole probe measurements. A schematic of the two-transducer setup is shown in Figure 3.6. For this setup a Scanivalve pressure stepping device is used to obtain the probe pressures. This type of data acquisition setup is slow, since after each Scanivalve step a certain amount of time must be allowed to ensure that pressure in the pneumatic circuit has reached the new value. The necessary waiting time had been established in previous studies (e.g. Rodger 1992). However, at the start of the present research some modifications were made to the instrumentation setup, and some pressure tubing was shortened or replaced. Therefore measurements of the pressure transients were performed. A typical pressure transient obtained for the three-hole probe, represented by the transducer output voltage, is shown in Figure 3.7. Waiting times of 25 seconds for the three-hole probe and 20 seconds for the seven-hole probe were found to be sufficient.

3.2.4 Four-Transducer Data Acquisition Setup

A schematic of the four-transducer data acquisition setup is shown in Figure 3.8. For this setup there is a pressure transducer dedicated to each hole of the three-hole probe, as well as the one dedicated to the upstream reference pressure difference.

Figure 3.6: Two-Transducer Data Acquisition Setup

Figure 3.7: Transients in Three-Hole Probe Measurements After Scanivalve Step (for probe incidence of 0°)

Figure 3.8: Four-Transducer Data Acquisition Setup

Since there are no pressure transients created by Scanivalve stepping no waiting time is needed, which greatly accelerates the data gathering procedure. Switching to the four-transducer setup required some minor changes to the data acquisition software, THP_TRV, and also required the wiring of a new multi-channelled switch in order to display the transducer readings on the digital voltmeter. Identical runs were performed using the two-transducer setup and the four-transducer setup to verify the repeatability of the results. All changes made to the data acquisition software are documented in the software's source code. Note that the four transducer setup is only suitable for use with the three-hole pressure probe. An equivalent setup for the seven-hole probe would require at least eight transducers, which is economically impractical at this time.

3.2.5 Probe Mounting and Traversing

Inlet Probe Mounting and Traversing

The shorter three-hole probe is used to measure flow properties upstream of the cascade blades. The probe is mounted in a holder socket which is attached to the end of a rod. The rod extends through a slot cut in the inside endwall and is connected to the traversing gear by means of an offset, as shown in Figure 3.9. The traversing gear allows the probe to be traversed along the cascade Y-axis (pitchwise direction) over a range of about 250mm. The inlet traverse plane is located 60% of an axial chord upstream of the leading edge of the cascade, as shown in Figure 3.10. The Z-axis coordinate (spanwise direction) is fixed by the length of the rod and was always 100mm (midspan). The probe was aligned to an angle of 0° to the horizontal using a small bubble level. The accuracy of the inlet probe alignment was about $\pm 1^\circ$.

When not in use the inlet probe was removed along with its mounting rod, and the slot in the inner endplate was taped over.

Figure 3.9: Inlet and Outlet Probe Mounting.

Figure 3.10: Inlet and Outlet Traverse Planes.

Outlet Probe Mounting and Traversing

Two probes were used to measure the outlet flow, the seven-hole probe and the longer three-hole probe. Both probes are mounted in a holder socket which is attached to the end of the support rod, as shown in Figure 3.9. The rod extends back through a slot in the endwall where it is attached to the traversing gear. The traversing gear has two degrees of freedom, allowing probe movement in both the pitchwise and spanwise directions. The traverse gear for spanwise motion was manufactured at the start of the present research.

The probe alignment was achieved using a bubble level with an attached adjustable protractor. The accuracy on the protractor was about $\pm 0.5^\circ$, allowing the probe to be aligned to about $\pm 1^\circ$. The alignment of the probe was regularly checked to make sure no rotation of the probe had occurred.

All outlet traverses were performed in the pitchwise direction at a particular spanwise location. The outlet traverse plane was located 40% of an axial chord downstream of the trailing edge of the cascade. During the course of the present research, some backlash in the pitchwise position of the probe was noticed. Therefore, before each traverse, care was taken to ensure that all slack in the vertical traverse gear had been eliminated. The spanwise and pitchwise location of the probe was measured, to about ± 0.5 mm using a steel rule, to confirm its position before each run commenced. Occasional corrections to the spanwise position were necessary.

3.3 Data Acquisition and Reduction

3.3.1 Data Acquisition Software

The data acquisition software used for the present research were modified versions of the THP_TRV software developed by Whitehouse (1993b). This software controls the spanwise and pitchwise probe positioning and data acquisition processes. It also reduces three-hole probe data and gives graphical presentations of pressure loss,

velocity, and flow angle.

Some modifications were made to THP_TRV during the present research. As discussed in Section 3.2.4, modifications were made to the program to permit data acquisition using four transducers instead of two. Also, it was discovered that there were minor errors affecting the calculation of mass averaged flow quantities, particularly the calculation of the mass averaged flow angle. These errors were corrected. To keep track of modifications and corrections to the data acquisition software, a version numbering system has been implemented. At the end of the present research, the version number was THP_TRV v2.31. Previous versions of the program, including the Pascal source code, are kept in the turbomachinery lab on floppy disk.

Hetu (1993) modified the original three-hole probe version of THP_TRV for use with the seven-hole probe. The functionality of the seven-hole probe version, called SHP_TRV, is basically the same as the three-hole probe version. An additional output file, for raw seven-hole probe data, must be specified. This file must be reduced using the seven-hole probe reduction program, SHP_DATA, described below.

3.3.2 Data Reduction Software

Three-Hole Probe Data Reduction

The THP_TRV program was used to reduce the raw three-hole probe data to values of flow yaw angle β , pressure loss $(P_o - P_{o_{in}})/q_{in}$, and velocity V . Although THP_TRV also provides routines for calculating mass-averaged flow quantities, new programs were implemented during the present research, to be more flexible. These programs were all implemented in the C programming language on a Sun Workstation, and use the reduced data files output from THP_TRV as input. They are:

- **thp2gnu:** Convert THP_TRV reduced data file, from a single traverse, into a format readable by the GNUPLOT plotting package. Only user specified flow quantities are converted.

- **thp2mass:** Process THP_TRV reduced data file, from a single pitchwise traverse, to calculate pitchwise mass-averaged flow quantities.
- **thp2stg:** Process THP_TRV reduced data files from a series of pitchwise traverses, representing a planar grid of data, to calculate the mass averaged flow quantities over the entire plane.

Each program contains a help screen that explains the usage of the program.

Seven-Hole Probe Data Reduction

The seven-hole probe data reduction program SHP_DATA, developed by Danias (1987) and Yaras (1990), was used to reduce the seven-hole probe data. This program uses the additional output file from SHP_TRV as the raw data file, and outputs values of flow angles α and β (pitch and yaw), pressure loss $(P_o - P_{s,m})/q_{in}$, dynamic pressure q/q_{in} , and velocities u/V , v/V , and w/V . For further flexibility, the Pascal source code for SHP_DATA was ported to a Sun Workstation, where it was translated into C using the Pascal-to-C converter p2c. A few prompts were removed in order to speed up the reduction process. As with the three-hole probe, some programs were implemented in C to manipulate and reduce the data further. These programs are:

- **shp2gnu:** Convert SHP_DATA reduced data file, from a single traverse, into a format readable by the GNUPLOT plotting package. Only user specified flow quantities are converted.
- **shp2mass:** Process SHP_DATA reduced data file, from a single pitchwise traverse, to calculate pitchwise mass-averaged flow quantities.
- **shp2stg:** Process SHP_DATA reduced data files from a series of pitchwise traverses, representing a planar grid of data, to calculate the mass averaged flow quantities over the entire plane.

Each program contains a help screen that explains the usage of the program.

3.4 Establishing Baseline Cascade Performance

The influence of the damage on the blade aerodynamics was established by comparison with the results for the baseline, undamaged blade. The baseline cascade performance was established for every incidence angle tested, before measurements with damaged blades commenced. The following measurements were made: inlet flow angle; outlet flow periodicity; axial velocity ratio; and width of the uniform low loss region. The test section control surfaces were adjusted until these characteristics were within specifications which have been established in the previous studies of Rodger (1992), Whitehouse (1993), and Abbott (1993).

3.4.1 Inlet Flow Angle

The inlet flow angle is measured using the short three-hole pressure probe. This probe is mounted as described in Section 3.2.5. The inlet flow angle should measure within $\pm 0.5^\circ$ of zero and not vary more than 0.5° across the front of the two central blade passages. A typical measurement which meets with this specification is shown in Figure 3.11. The inlet flow angle is controlled using mainly the top and bottom flap control surfaces. It was found that, in general, the inlet flow angle was somewhat dependent on changes in both the tailboard positions and the endplate positions. Therefore the inlet flow angle should be checked after adjustments to any of these control surfaces.

3.4.2 Outlet Flow Periodicity

Ideally, the number of blades in a cascade should be infinity, or at least a very large number. This is obviously impractical. In this research, the presence of an infinite cascade is simulated by ensuring periodicity of the three center wakes of the five blade cascade. For periodicity to exist, the pressure loss measurements should appear similar through all three wakes. If the maximum pressure losses in each wake are

Figure 3.11: Acceptable Inlet Flow Angle Measurement (undamaged, $i = -10^\circ$).

compared, the largest maximum and the smallest maximum should not differ by more than 0.01. Also, the flow angles measured in the two adjacent blade passages should be similar. The flow angles measured at $Y/S = 0.5$ and $Y/S = -0.5$ should not differ by more than 1° . Acceptable periodicity measurements, obtained for an incidence of 0° , are shown in Figures 3.12 and 3.13. The small positive losses observed in the free stream imply the measured stagnation pressure is lower upstream of the cascade than it is downstream. This may be due to the fact that the measurements shown are from a quick traverse, with very short sample periods. It is also possible that there are problems with the upstream stagnation pressure probe, either with the probe placement (inside the boundary layer), or with the probe itself. However, this error can be corrected for, as described in Section 5.1.2.

The outlet flow angles and the pressure losses are controlled by adjusting the top and bottom tailboards of the cascade test section. During the present research it was found that it was not always possible to satisfy both pressure loss and flow angle periodicity criteria at once. At positive incidence, the wake at $Y/S = -1.0$ has consistently higher losses than the other wakes. Therefore a compromise had to be made between pressure loss and flow angle periodicity.

3.4.3 Axial Velocity Ratio

The axial velocity ratio (AVR) can be defined mathematically as

$$AVR = \frac{V_{axial_{te}}}{V_{axial_{le}}} \quad (3.1)$$

where $V_{axial_{te}}$ and $V_{axial_{le}}$ are mass-averaged as

$$V_{axial_{te}} = \frac{\int V_{axial} d\dot{m}}{\int d\dot{m}} \bigg|_{te} \quad (3.2)$$

and

$$V_{axial_{le}} = \frac{\int V_{axial} d\dot{m}}{\int d\dot{m}} \bigg|_{le} \quad (3.3)$$

Figure 3.12: Acceptable Pressure Loss Periodicity Measurements (undamaged, $i = 0^\circ$).

Figure 3.13: Acceptable Flow Angle Periodicity Measurements
(undamaged, $i = 0^\circ$).

The leading edge and trailing edge quantities are measured at midspan. An *AVR* equal to 1.0 is a necessary, but not sufficient, criterion for ensuring two-dimensional flow through the cascade.

In the present research, $V_{axial_{40}}$ and $V_{axial_{60}}$ are actually interpolated from the mass averaged-axial velocities measured 40% of an axial chord upstream and 60% of an axial chord downstream. The interpolation is made assuming a linear variation of mass averaged axial velocity between the upstream and downstream measurement planes. The *AVR* was set by adjusting the amount of divergence until an *AVR* = 1.0 ± 0.02 was achieved.

3.4.4 Width of Uniform Low Loss Region

Another necessary (but again not sufficient) criterion for establishing two-dimensional flow through the cascade is the existence of a region of uniform losses around the midspan of the blade, as shown in Figure 3.14. Care was taken to ensure that the width of the uniform loss region was wider than the extent of the 25% damage notch, the largest damage size tested, so that the effects of the damage could be compared with the baseline performance without having to account for the endwall effects.

If the uniform inlet flow and outlet periodicity have been achieved, then a narrow uniform loss region is usually the result of an ineffective seal between the cascade blades and the endplates. If leakage occurs, the air from outside of the endplates leaks into the passage, apparently causing boundary layer separation in the corner between the suction surface and the endwall. Another cause for a narrow uniform loss region could be separation at the leading edge of the endplates, since this thickens the boundary layer on the endplate, causing larger secondary flows. The flow over leading edges of the endplates can be checked using surface oil flow visualization. If separation is detected, the flexible leading edge of the endplate should be a...

The midspan losses measured during the present research compared well with results obtained in previous studies by Rodger (1992), Whitehouse (1993), Klein

Figure 3.14: Spanwise Loss Distribution Showing Uniform Loss Region (undamaged, $i = 0^\circ$).

Figure 3.15: Spanwise Pressure Loss Distribution (25% damage, $i = 0^\circ$).

(1993), and Abbott (1993).

3.5 Measuring Damaged Blade Performance

3.5.1 Establishing Spanwise Symmetry of Downstream Flow

Figure 3.15 shows the spanwise pressure loss distribution measured for the 25% damaged blade at $i = 0^\circ$. The flow symmetry about the midspan is fairly good, but not perfect. As can be seen in Figure 3.14, which shows a spanwise pressure distribution of total pressure loss for the baseline case, the uniform loss region is not centered about the midspan. Thus exact symmetry should not be expected for the damaged case. However, the symmetry observed was considered acceptable.

Therefore for subsequent tests only the positive side of the span was measured, since this is the side on which the baseline blade showed the widest region of uniform losses. By exploiting this approximate symmetry, considerable time was saved during the measurements.

3.5.2 Establishing Magnitude of Downstream Flow Angularity

From Klein (1993) it was known that the cutout generated two counter-rotating vortices in the flow downstream of the cascade. Since the strength of these vortices was not known, the seven-hole probe was used for the initial measurements, which were performed for 25% damage at 0° incidence. Figures 3.16 and 3.17 show the pitch and yaw angles measured for pitchwise traverses at various spanwise stations (measured in mm from the center of the cutout). The measured pitch and yaw angles never exceeds 7° in magnitude, relative to the long axis of the probe. The three-hole probe is accurate for up to $\pm 10^\circ$ of pitch misalignment. Therefore subsequent measurements used the three-hole probe instead of the seven-hole probe, to speed up the data gathering procedure. Subsequent tests showed that the flow angularity does not increase significantly with either incidence angle or cutout size.

Figure 3.16: Pitch Angle Measured Downstream of Damaged Blade
(25% Damage, $i = 0^\circ$)

**Figure 3.17: Yaw Angle Measured Downstream of Damaged Blade
(25% Damage, $i = 0^\circ$)**

Chapter 4

Computational Analysis

The Dawes code was used to model the flow through the linear cascade of turbine blades, used in the experimental part of the present research, as closely as possible. Detailed information concerning the geometry and flow conditions in the experimental cascade is contained in Chapter 3. The objective was to investigate how well the Dawes code could predict the flows, for both the undamaged and damaged cascades. This chapter describes how the Dawes code was used in the present research. The first section describes how the behaviour of the Dawes code was established for an undamaged cascade, for factors such as grid density, number of time steps, and inlet and exit Mach numbers. The following section describes how the computational grid was modified to model trailing edge damage. The final sections briefly describe the pre- and post-processors that were developed to analyze the Dawes code solutions.

4.1 Establishing Behaviour of the Dawes Code for an Undamaged Cascade

Before the Dawes code was used to predict the flow through a damaged cascade, the behaviour of the code for an undamaged cascade was examined. Factors such as the number of calculation time-steps, the grid density, and the inlet and outlet Mach numbers can have a significant impact on the stability and reliability of the solution. This section describes what tests were performed to establish the behaviour of the Dawes code, and what conclusions were derived from these tests.

4.1.1 Number of Time-Steps to Steady-State Conditions

The Dawes code requires the user to specify the number of time-steps for which a solution is to be calculated. For consistency's sake, it was desired that all Dawes code solutions used in the present research be calculated to the same number of time-steps. Obviously, it is very important to establish how many time-steps are necessary to achieve a steady-state (i.e. converged) solution. Specifying too few time-steps will result in an unconverged solution. Specifying too many time-steps is wasteful of time and computational resources.

The Dawes code supplies a measure of convergence in the form of a calculated *residue*. The residue is essentially a measure of the average imbalance of mass, momentum, and energy for each finite volume cell over the entire computational grid. As a steady-state solution is approached, the residue should approach zero. Figure 4.1 shows a typical variation of the residue over 1500 time-steps. Using the left scale, the residue appears constant after about 1000 time-steps. However, using the expanded right scale, the residue appears to still be falling at 1500 time-steps. Since the residue alone does not give much information concerning the appearance of the solution, it is difficult to judge when an "acceptable" steady-state has been reached.

Figure 4.1: Residue vs. Time-Steps (undamaged, $i = 0^\circ$, $M_{te} = 0.70$).

Figure 4.2: Pitchwise Mass-Averaged Total Pressure Loss Distribution at Various Time-Steps (undamaged, $i = 0^\circ$, $M_{te} = 0.70$).

An alternative approach is to examine the trend of a specific quantity, such as the total pressure loss which is of primary interest in the present research. For most of the present research Dawes code solutions were calculated to 1500 time-steps, with solutions being stored at 500, 750, 1000, 1250, and 1500 time-steps. Spanwise distributions of pitchwise mass-averaged total pressure losses were compared at the different time-steps. An example is shown in Figure 4.2. This solution represents an undamaged subsonic cascade, and utilizes a grid with $I_{max} \times J_{max} \times K_{max}$ dimensions of $33 \times 107 \times 33$. The results have been extracted from a plane 40% of an axial chord downstream of the trailing edge of the cascade. The endwall regions of the solution have been excluded, for reasons discussed in Section 4.1.2. The total pressure loss appears to change little after about 1250 time-steps. This behaviour was consistent for

all cases examined. The computational results presented in Chapter 5 were therefore calculated to 1250 time-steps.

4.1.2 Uniform Flow Region

For the present research, it was desired to isolate the effects of trailing edge damage from the flows associated with endwall boundary layers. For the computational portion of the research, it was hoped to achieve this by using the inviscid endwall model provided by the Dawes code. Unfortunately, early tests showed that the inviscid endwall model was not properly implemented in the Dawes code. Dawes subsequently confirmed that this was the case. Negative losses were predicted at the endwalls. Eventually these "losses" would propagate away from the endwalls to affect the entire solution. As a result, the inviscid endwall model was not used.

Utilizing the viscous endwall model introduced endwall boundary layers, and their associated vortical flows, to the solution. To move these endwall effects away from the midspan region, the aspect ratio of the cascade was increased. This left a wide uniform loss region about the midspan of the cascade. When analyzing a solution, the endwall effects could be removed by omitting the finite volume cells close to the endwalls. This is illustrated schematically in Figure 4.3, which shows a spanwise distribution of total pressure loss including the endwall regions. Virtual "hub" and "case" locations are chosen, outside of the endwall regions. The spanwise locations are normalized by the span between the virtual hub and case, such that $Z/H = 0$ coincides with the virtual hub, and $Z/H = 1$ coincides with the virtual case. When analyzing a solution, only the region $0.0 < Z/H < 1.0$ is considered. The endwall regions, $Z/H < 0.0$ and $Z/H > 1.0$, are omitted. The computational results presented in Chapter 5 were calculated using an aspect ratio of 1.97, and analyzed using an aspect ratio of about 1.0.

Since the endwall regions are omitted, it is no longer necessary to cluster grid nodes close to the endwalls in order to accurately capture the endwall boundary

Figure 4.3: Pitchwise Mass-Averaged Total Pressure Loss Distribution, Showing Excluded Endwall Region (undamaged, $i = 0^\circ$, $M_{te} = 0.70$).

Table 4.1: Summary of Grid Density Runs.

Grid Dimensions $I_{max} \times J_{max} \times K_{max}$	Number of Grid Nodes	Output File Size (kB)	Computation Time 1500 time steps (hrs:min:sec)	Mass Averaged Total Pressure Loss, $\bar{Y}$
17x107x17	30923	2999	5:17:56	0.050018
25x107x25	66875	6615	11:48:51	0.042167
33x107x33	116523	11641	20:49:05	0.028981
37x107x37	146483	14684	25:56:52	0.033323

layers. This allows a higher grid density towards the midspan of the blade, which is useful when resolving the flow through a cascade with trailing edge damage.

4.1.3 Effects of Grid Density on Undamaged Cascade

In the present research, 107 axial grid points were used in all runs. To test the dependence of the solution on the spanwise and pitchwise grid density, solutions were computed with grids having $I_{max} \times J_{max} \times K_{max}$ dimensions (i.e pitchwise $\times$ streamwise $\times$ spanwise dimensions) of 17 $\times$ 107 $\times$ 17, 25 $\times$ 107 $\times$ 25, 33 $\times$ 107 $\times$ 33, and 37 $\times$ 107 $\times$ 37. The solutions were calculated for the design incidence, for a subsonic cascade. Figure 4.4 shows spanwise distributions of mass-averaged total pressure loss for the various grid densities, after 1250 timesteps. Corresponding losses, measured in the cascade wind tunnel, are included for comparison. The results obtained for the 33 $\times$ 107 $\times$ 33 and 37 $\times$ 107 $\times$ 37 are in quite good agreement with the measured loss, while the losses for the lower grid densities are significantly higher. In terms of computation time and disk storage, the 33 $\times$ 107 $\times$ 33 grid is more attractive than the 37 $\times$ 107 $\times$ 37 grid, as is shown in Table 4.1. The computational results presented in Chapter 5, for both undamaged and damaged cascades, were calculated using grids with dimensions 33 $\times$ 107 $\times$ 33.

Figure 4.4: Pitchwise Mass-Averaged Total Pressure Loss Distribution for Various Grid Densities, after 1250 Time-Steps (undamaged, $i = 0^\circ$, $M_{te} = 0.70$).

Figure 4.5: Leading and Trailing Edge Mach Number vs. Inlet Total Pressure (undamaged, $i = 0^\circ$).

4.1.4 Effects of Increased and Decreased Cascade Mach Number

Control of Cascade Mach Number

The inlet and outlet Mach numbers of the cascade can be controlled by changing the specified inlet total pressure and outlet static pressure, essentially changing the pressure drop across the cascade. A number of test cases were calculated to determine the range of Mach numbers for which the Dawes code could remain stable. For these test cases, the outlet static pressure was kept constant at $71448 Pa$, and the inlet total pressure was varied. A grid of dimensions $33 \times 107 \times 33$ was used for all of the runs. Figure 4.5 shows how the leading edge Mach number (M_{le}) and the trailing edge Mach number (M_{te}) varies with the inlet total pressure ($P_{0,in}$). No stability problems were

encountered up to an inlet total pressure of 220000 Pa ($M_{te} = 0.46$, $M_{te} = 1.52$), the highest pressure solution attempted. A stable solution was achieved for a run with the inlet total pressure as low as 80000 Pa ($M_{te} = 0.29$, $M_{te} = 0.41$). However, as discussed below, the predicted solution in this case did not appear realistic.

Shockwave Capture for Transonic Cascade

Figures 4.6 and 4.7 show contours of constant Mach number, for M_{te} of about 0.70 and 1.14, respectively. For the latter case, the characteristic shock and expansion waves at the trailing edge of the cascade are clearly visible. The maximum predicted Mach number occurs near the suction surface, close to the trailing edge of the blade, and is about 1.53. The maximum Mach number in the subsonic cascade is 0.86, and is located near the suction surface but farther from trailing edge of the blade.

The shock waves captured by the Dawes code appear to be smeared out over several finite volume cells, which is to be expected. To see if more precise shock wave capture was possible using a denser grid, a transonic solution was calculated using a grid with dimensions $37 \times 107 \times 37$. It was found that the shock wave, while still somewhat smeared out, was more precisely captured using the denser grid. However, a comparison of the mass averaged total pressure losses for the two grids showed little difference. Therefore, to save time and disk storage space, all transonic runs were performed using the same $33 \times 107 \times 33$ grid used for the subsonic runs.

Total Pressure Loss vs. Mach Number

Figure 4.8 shows the predicted mass-averaged total pressure loss as a function of the predicted exit Mach number. Included in the figure are predictions of the correlations of Ainley and Mathieson (1957) and Kacker and Okapuu (1981), for the same blade geometry. In the range $0.60 < M_{te} < 1.0$, Kacker and Okapuu is in very good agreement with the Dawes code predictions. Ainley and Mathieson predicts somewhat higher losses, which is not surprising since the correlation corresponds to older, less

Figure 4.6: Contours of Constant Mach Number (undamaged, $i = 0^\circ$, $M_{te} = 0.70$).

Figure 4.7: Contours of Constant Mach Number (undamaged, $i = 0^\circ$, $M_{te} = 1.14$).

Figure 4.8: Mass-Averaged Total Pressure Loss vs. Trailing Edge Mach Number (undamaged, $i = 0^\circ$).

efficient turbine designs. For $M_{te} < 0.60$, the Dawes code predictions were found to be unrealistic. Negative losses were being predicted, even though the solutions remained stable. This behaviour is not very surprising, since the Dawes code does not claim to be applicable for very low Mach numbers. For $M_{te} > 1.0$ the Dawes code predicts a sharp rise in the losses, which is expected due to the additional shock losses. Ainley and Mathieson shows a sharper rise in losses for $M_{te} > 1.0$ than the Dawes code calculations. The Kacker and Ckapuu correlation was not suitable for this range.

4.2 Development of Grid Modifications to Simulate Trailing Edge Damage

In the present research, trailing edge damage was modelled by adding a semi-circular cutout at the midspan of the trailing edge. The following sections describe the grid modifications necessary to model this cutout with the Dawes code. The computational grid for the undamaged cascade will be referred to as an *undamaged grid*, and the computational grid for the damaged cascade as a *damaged grid*.

4.2.1 Generating a Damaged Grid Description

To describe a blade profile to the Dawes code, a cylindrical $(x, r, r\theta)$ coordinate system is used. The description is illustrated schematically in Figure 4.9. The line of J_{max} nodes represents a single streamwise line on the meridional surface that is to coincide with the suction surface of the blade, as shown in Figure 4.9 (a). The profile is described by specifying the coordinates $(x, r, r\theta)$, and the blade thickness $\Delta(r\theta)$, for each node, as shown in Figure 4.9 (b). For all nodes with $J_{te} > J > J_{te}$ (i.e. all nodes not on the blade surface), the blade thickness is specified as $\Delta(r\theta) = 0.0$.

To specify the grid for a simple blade row where the geometry varies linearly across the blade span, the Dawes code only requires a description of the blade profile at the

(a)

(b)

Figure 4.9: Schematic of the Blade Profile Description Used by the Dawes Code:
 (a) meridional surface coinciding with blade suction surface; (b) streamwise
 line of nodes from meridional surface.

hub and at the casing. The Dawes code interpolates the profiles at intermediate surfaces between the hub and the case. This will not work for a damaged blade, since the blade profile changes significantly inside the damage. To specify damaged blade geometry, it is necessary to describe the blade profile for every streamwise surface between the hub and the case. For streamwise surfaces outside the damage, the undamaged profile description can be used. For streamwise surfaces inside the damage, a damaged profile description is generated by modifying the undamaged profile description.

Modifying the undamaged profile description involves redistributing the nodes to fit the damaged profile. The Dawes code requires that J_{te} be the same for every streamwise surface of the grid. Since the damaged profile is essentially a truncated undamaged profile, the J_{te} node must be shifted to the new location of the trailing edge, with a corresponding stretching of the rest of the grid, as shown in Figure 4.10.

4.2.2 Modifications of Trailing Edge Geometry

At the beginning of the present research, the damaged profile descriptions set the blade thickness inside the damage abruptly to zero at the J_{te} node, as shown in Figure 4.11 (a). This abrupt change in geometry led to stability problems, encountered for situations where the nodes around the trailing edge were very closely spaced in the axial direction. Rounding of the trailing edge at the pressure-side corner, as shown in Figure 4.11 (b), appears to have solved these stability problems. The rounded trailing edge is still a reasonable approximation of the geometry of the damaged blades used in the experimental research, as can be seen looking at the actual physical damaged blade profile shown in Figure 4.11 (c).

It was desired to distribute as many streamwise surfaces across the span of the cutout as possible, to increase the resolution of the flow in the cutout region. However, it was found that if more than five streamwise surfaces were used across the cutout, the

Figure 4.10: Damaged Blade Profile Description.

(a)

(b)

(c)

Figure 4.11: Finite Volume Cells Near Damaged Trailing Edge, Viewed from Streamwise Surface at Midspan (Center of Cutout): (a) sharp trailing edge; (b) rounded trailing edge; (c) physical damaged blade profile.

Figure 4.12: Finite Volume Cells Near Damaged Trailing Edge, Viewed from Meridional Surface: (a) 5 streamwise surfaces across cutout; (b) 9 streamwise surfaces across cutout.

Dawes code became unstable within the first few time-steps. A possible explanation for this may be the fact that some of the finite volume cells are skewed such that their "upstream" faces are nearly parallel to the local velocity vector. The situation worsens as more streamwise surfaces are used across the cutout, as shown in Figure 4.12. To correct this problem, the top and bottom edges of the cutout were rounded by applying a radius equal to one-half the radius of the cutout itself, as shown in Figure 4.13. Using this modification it was possible to place up to 13 grid nodes across the cutout. Subsonic solutions were calculated for grids with 5, 9, 11, and 13 nodes distributed across a 25% cutout. For the first three cases, a grid with dimensions $33 \times 107 \times 33$ was used. For the last case, grid dimensions of $33 \times 107 \times 37$ were necessary. The results for the pitchwise mass-averaged loss coefficients are shown in Figure 4.14. The predicted spanwise distribution varies little for 9 or more nodes across the cutout. The loss peaks at the edges of the cutout are of approximately the

diagram not to scale

Figure 4.13: Rounding of the Top and Bottom Edges of Cutout.

Figure 4.14: Mass-Averaged Total Pressure Loss Distribution for Various Numbers of Streamwise Surfaces Across Cutout (25% damage, $i = 0^\circ$, $M_{te} = 0.70$).

correct magnitude, compared with experimental results (See Figure 3.15). However, the negative losses at the centre of the cutout are not physically realistic, since these represent a small gain in total pressure. The reasons for these pressure gains are unclear, though some possibilities are mentioned in Chapter 5.

The damaged cascade results presented in Chapter 5 were calculated using 9 streamwise surfaces across the cutout.

Figure 4.15 shows streamwise and meridional surfaces for a grid with dimensions $33 \times 107 \times 33$, which incorporates all of the trailing edge modifications described above. The streamwise surface cuts the blade midway through the cutout. This grid was used to calculate all of the damaged results presented in Chapter 5.

4.3 Pre- and Post-Processing Software

4.3.1 The CASTLE Preprocessor

Generating a Dawes code input file that describes the grid geometry, and boundary and initial conditions, is not a straightforward procedure. This is especially so for a damaged grid input file, which can be over 2000 lines in length. Thus there was a need for a pre-processor to partially automate the grid generation process. To meet this need, the program CASTLE was developed.

CASTLE allows the user to generate a Dawes code input file. Using coordinates describing the cascade blade profile as a basis, CASTLE allows the user to control such geometric parameters as the cascade stagger angle, the grid dimensions, and the grid clustering, and also the solution boundary and initial conditions. It also allows the user to add trailing edge damage of any size, with rounding of the trailing edge and cutout corners if desired.

The program CASTLE was implemented in FORTRAN on a Sun workstation. Instructions for its use are contained in Appendix B.

Figure 4.15: Computational Grid ($33 \times 107 \times 33$) for Trailing Edge Damaged Blade: (top) streamwise surface at midspan; (bottom) meridional surface.

4.3.2 Post-Processors: Extract and Centrepont

In order to analyze the solutions, some sort of post-processing software is essential. The Dawes code stores with each solution the values of the axial velocity, radial velocity, tangential velocity, relative Mach number, static pressure, density, and viscosity for each finite volume cell of the computational grid. A post-processor should extract all these values from a Dawes code solution file, and calculate quantities of interest such as mass-averaged total pressure loss coefficients and trailing edge deviation.

With the Dawes code, post-processing is complicated by the fact that the solution values are associated with the centres of the finite volume cells, not the known corner nodes. Two approaches for analyzing the data were investigated: the flow values can be interpolated to the coordinates of the cell corners (program Extract); or the coordinates of the cell centres can be calculated (program Centrepont).

The Program Extract

The program Extract analyzes a Dawes code solution by interpolating the stored flow data to the coordinates of the cell corners. This interpolation is performed by simply averaging the flow values from the eight cells surrounding each node. Extract is able to calculate flow values for any streamwise, meridional, and quasi-orthogonal surface. The flow values can be mass-averaged over an entire surface, or saved as an array of data from which a contour plot can be generated. For quasi-orthogonal surfaces, the flow values can be mass-averaged in the pitchwise direction, to produce a spanwise distribution of the data. In addition to the flow values stored in a Dawes code solution file, the total loss pressure loss, Y and C_{p0} , is also available. The program Extract was implemented in FORTRAN on a Sun workstation.

A consequence of this simple interpolation method is that the interpolation process appears to inherently smooth the solution. As a result, minor numerical instabilities, such as the "sawtooth" contours seen in Figures 4.6 and 4.7, can be completely hidden.

Also, shock waves that occur in transonic flows appear further smeared out. Since, in most circumstances, this sort of smoothing is not desirable, the program Centrepoint was developed.

The Program Centrepoint

The program Centrepoint analyzes a Dawes code solution by calculating the coordinates of the centre of every finite volume cell of the computational grid. Since the geometry of a finite volume cell is, in general, arbitrary, a weighted averaging process is used to calculate the coordinates of the cell centre. The finite volume cell is divided into a number of planes. The centroid and area of each plane is calculated, assuming that the planes are two-dimensional trapezoids. The coordinates of the centroids are then averaged, weighted with their associated areas, to get the coordinates of the centre of the finite volume cell.

The program Centrepoint allows the user to extract the flow values from any streamwise, meridional, and quasi-orthogonal plane. The values can be saved as an array of data from which a contour plot can be generated. For quasi-orthogonal surfaces, the values can be mass-averaged over the entire surface, or mass-averaged in the pitchwise direction only to produce a spanwise distribution of the data. A specified number of endwall nodes can be excluded from the mass-averaging calculation, so that endwall effects are not included. In addition to the flow values stored in a Dawes code solution file, the total pressure loss coefficients, Y and C_{p0} , the flow angle, and the trailing edge deviation are also available. The program Centrepoint was implemented in FORTRAN on a Sun workstation, and was the post-processor used to extract the computational results presented in Chapter 5.

PRECISIONSM RESOLUTION TARGETS

Chapter 5

Experimental and Computational Results

5.1 Experimental Results

5.1.1 Measured Wake Profiles

Figure 5.1 shows the total pressure losses measured by traversing the wake downstream of a damaged blade, at various spanwise locations. The results are for incidences of $+10^\circ$ and -10° for the 25% and 15% cutouts. The spanwise location of each traverse is measured from the center of the cutout, and is normalized by the radius of the cutout, R_d . For comparison, midspan measurements for the undamaged blade at the same incidence are included. The wake is seen to be altered considerably by the presence of the cutout. At the centre of the cutout ($Z/R_d = 0$) the losses actually appear to be lower, while the wake is considerably broadened at the edge of the cutout ($Z/R_d = 1.0$). There is also considerable pitchwise displacement of the wakes, suggesting more under turning of the passage flow than would be expected based on the chordwise extent of the cutout. The double peaked wake profiles observed near the edges of the cutout are an unexpected result. The double peaks are most prominent for the 25% cutout, and were observed at all incidences tested. The reasons for the double peaked profiles are not known.

Figure 5.1: Total Pressure Losses Measured in Damaged Blade Wakes:
 (a) 15% Cutout, $i = -10^\circ$; (b) 25% Cutout, $i = -10^\circ$;
 (c) 15% Cutout, $i = +10^\circ$; (d) 25% Cutout, $i = +10^\circ$.

5.1.2 Spanwise Distribution of Pressure Loss and Deviation

Based on traverse measurements such as those in Figure 5.1, pitchwise mass-averaged results were calculated according to:

$$X' = \frac{\int_{-0.5}^{0.5} X V_x d(\frac{Y}{S})}{\int_{-0.5}^{0.5} V_x d(\frac{Y}{S})} \quad (5.1)$$

where X is the flow value being mass-averaged (e.g. total pressure loss, trailing edge deviation, velocity). When calculating the mass-averaged pressure loss, the effects of the small positive losses observed in the free stream (see Figure 5.1) are negated by shifting the traverse measurements such that the average of the free stream losses is zero. To do this, a linear fit is performed on the losses measured in the free stream. This line is subtracted from the losses measured at every point of the traverse. Only then is equation 5.1 used to calculate the mass-averaged loss.

Figure 5.2 shows pitchwise mass-averaged results of total pressure loss, presented for incidences of -10° , 0° , $+5^\circ$, and $+10^\circ$ for both 15% and 25% cutouts. The extent of the 15% and 25% damage and the pitchwise mass-averaged results for the undamaged blade are included for reference. These measurements support the conclusion of Klein (1993) that counter-rotating streamwise vortices are shed from the edges of the cutout. An analogy can be drawn from tip leakage flows, since the edges of the cutout form swept leakage paths from the pressure surface to the suction surface. As the leakage flow interacts with the passage flow it would be expected to roll up into two counter-rotating "edge" vortices. Regions of high loss, of the sort observed for other vortical flows, are seen to coincide with the edges of the cutouts.

The presence of edge vortices would cause significant under turning of the flow around the middle of the cutout. This under turning of the flow is observed in the pitchwise mass-averaged results of trailing edge deviation, $\delta = \alpha_2 - \beta_2$, shown in Figure 5.3, and was also suggested by the wake profiles shown in Figure 5.1. Edge vortices would also cause over turning beyond the extent of the cutouts, which is also

Figure 5.2: Pitchwise Mass-Averaged Total Pressure Loss for Various Incidence Angles.

observed in Figure 5.3.

One rather surprising observation, already mentioned in connection with Figure 5.1, is that the total pressure losses downstream of the centre of the cutout are, in most cases, actually lower than the undamaged losses. This is particularly noticeable at the higher incidences. The boundary layers would be expected to be thinner at the cutout trailing edge, since the blade surface length is shorter than to the undamaged trailing edge. However, the fact that the lower losses are also observed beyond the edges of the cutout suggests a further explanation is necessary. It may be that the boundary layer fluid approaching the cutout is swept into the edge vortices, effectively thinning the boundary layers reducing the accumulated losses in this region. The boundary layer fluid swept into the edge vortices would contribute to the higher losses observed downstream of the edges of the cutout, adding to the extra losses associated with the vortical flows. It is also possible that the cutout flow may be altering the base pressure in such a way as to reduce the trailing edge contribution to the measured losses inside of and beyond the edges of the cutout. Further insight into the mechanisms at work in the damaged cascade would be gained using oil visualization on the blade surfaces. This was not done, and remains work for future studies.

5.1.3 Mass-Averaged Results

To provide a measure of the performance of the entire blade row, the traverse measurements were mass-averaged over the entire downstream measurement plane according to:

$$\bar{X} = \frac{\int_{-0.5}^{0.5} \int_{0.0}^{0.5} X V_x d\left(\frac{Z}{H}\right) d\left(\frac{Y}{S}\right)}{\int_{-0.5}^{0.5} \int_{0.0}^{0.5} V_x d\left(\frac{Z}{H}\right) d\left(\frac{Y}{S}\right)} \quad (5.2)$$

where X is the flow value being mass-averaged (e.g. total pressure loss, trailing edge deviation, velocity). Z is measured from midspan, and H was set equal to the blade chord. Thus, the mass-averaged results represent those for a hypothetical blade row

Figure 5.3: Pitchwise Mass-Averaged Trailing Edge Deviation for Various Incidence Angles.

Figure 5.4: Mass-Averaged Total Pressure Loss for Blade with Unit Aspect Ratio.

of unit aspect ratio, assuming symmetry about the blade midspan. Endwall effects were omitted by replacing the outer measurements with appropriate measurements from the uniform loss region of the undamaged blade.

Figure 5.4 shows the mass-averaged total pressure losses which would be observed for a blade with unit aspect ratio. It is evident that the change in losses for the blade, as a function of cutout diameter, is small. In fact small decreases in losses were observed for $i = +10^\circ$. Though the range of results is of the order of the accuracy of the measurements, suggesting the observed trends should be treated with caution, it is interesting to recall the study by DeCook et al. (1993) reviewed in Section 2.5. In a study of compressor blades with serrated trailing edges, DeCook et al measured a similar small but noticeable reduction in the total pressure losses.

5.2 Computational Results

The objectives of the computational part of the present research was to evaluate the ability of the Dawes code to predict the flow through a cascade of turbine blades with trailing edge cutouts. Therefore, in the following sections, there will be more emphasis placed on the qualitative aspects of the computational results rather than the quantitative aspects. Where possible, experimental results will be presented with the computational results for comparison.

5.2.1 Overview of Calculated Flow Field

Figures 5.5 and 5.6 show contours of constant Mach number on streamwise surfaces cutting the flow field through the centre of the cutout (i.e. blade midspan) and away from the cutout, respectively. The presence of the cutout significantly alters the wake. There is considerably more under turning of the flow at the centre of the cutout, and the wake appears to become mixed-out with the main passage flow much more rapidly than away from the cutout. The considerable under turning suggests that the formation of the edge vortices, observed in the experimental results, is being reproduced by the Dawes code. Whether the rapid mixing is also an effect of the edge vortices is not clear, though some studies (e.g. DeCook et al, 1993) have suggested this to be the case.

5.2.2 Spanwise Distribution of Pressure Loss and Deviation

Figures 5.7 and 5.8 show the pitchwise mass averaged results of total pressure loss and trailing edge deviation, respectively. The results have been extracted from a plane 40% of an axial chord downstream of the trailing edge of the cascade. The losses are presented for a cutout diameter of 25% chord, at incidences of -10° , 0° , $+10^\circ$. The calculated undamaged losses, and the experimental damaged and undamaged losses, are included for comparison.

Figure 5.5: Contours of Constant Mach Number at Midspan (25% damaged, $i = 0^\circ$, $M_{te} = 0.70$).

Figure 5.6: Contours of Constant Mach Number at $Z/R_d \approx 4$ (25% damaged, $i = 0^\circ$, $M_{te} = 0.70$).

Figure 5.7: Pitchwise Mass-Averaged Total Pressure Loss for Various Incidence Angles: Dawes Code and Measured Results.

Figure 5.8: Pitchwise Mass-Averaged Trailing Edge Deviation for Various Incidence Angles: Dawes Code and Measured Results.

Many of the flow characteristics are well predicted. The calculated loss maxima near the edges of the cutout are generally in good agreement with the measured results. The fact that the lateral extent of the loss peaks are similar to those measured suggests that the diffusion that occurs in the flow downstream of the cutout is well represented. The Dawes code does not predict a drop in losses beyond the edges of the cutout, as was observed in the experimental measurements. Also, the total pressure gains predicted inside the cutout are obviously unrealistic. The reasons for the total pressure gains are not clear. Since a small reduction of losses at the centre of the cutout was measured in the experimental cascade, it is possible that the mechanisms leading to this reduction are being over predicted, leading to the pressure gains.

The Dawes code over predicts the amount of under turning that exists in the region of the cutout, when compared with the measured trailing edge deviation. Both the maximum deviation and the spanwise extent of the under turned region are over estimated. Nevertheless, the Dawes code does seem to capture many of the essential features of the flow.

5.2.3 Mass-Averaged Results

Figure 5.9 shows the mass-averaged total pressure loss corresponding to a blade of unit aspect ratio. The magnitude of the losses are very similar to the measured results shown in Figure 5.4. The Dawes code predicts a slight rise in total pressure losses with increasing cutout size, as opposed to the slight decrease seen with some of the measured results. However, as mentioned in Section 5.1.3, the inherent uncertainties in the measured mass averaged results suggest that observed trends be treated with caution. In any case, the Dawes code shows that the effects of trailing edge damage on mass averaged losses is weak, in agreement with the measured results.

Figure 5.9: Dawes Code Predictions of Mass-Averaged Total Pressure Loss for a Blade with Unit Aspect Ratio.

5.2.4 Effects of Increased Mach Number

While it is hoped that, qualitatively at least, the essential features of the cascade flows are being captured by the transonic Dawes code solutions, there are at present no corresponding experimental measurements of transonic cascades with trailing edge cutouts. Such experimental data is necessary before any reliable conclusions can be drawn from the following results.

Mach Number Effects on Overall Flow Field

Figures 5.10 and 5.11 show contours of constant Mach number on streamwise surfaces cutting the flow field through the centre of the cutout (i.e. blade midspan) and away from the cutout, respectively. The Mach number at the trailing edge of the cascade is $M_{te} = 1.14$. As observed in the subsonic cascade contour diagrams shown in Figures 5.5 and 5.6, there is considerable under turning visible at the centre of the cutout in comparison to away from the cutout. This suggests that the edge vortices are still being predicted in the transonic range. The shock wave that forms at the trailing edge of the blade appears more smeared out and weaker at the centre of the cutout than away from the cutout. It is not clear whether this is an actual physical effect (e.g. perhaps the local Mach number is lower near the cutout) or a numerical effect caused by the distortion of the grid around the cutout.

Mach Number Effects on Spanwise Distributions

Figures 5.12 and 5.13 show pitchwise mass-averaged total pressure loss and trailing edge deviation for trailing edge Mach numbers of $M_{te} = 1.07$ and $M_{te} = 1.14$. The results shown are for design incidence, which are similar to the results obtained for off-design ($i = +10^\circ, -10^\circ$) incidences. The loss distributions show many of the same features as observed with the subsonic cascade (See Figure 5.7), with some minor differences. There is a region of low losses beyond the edges of the cutout, which was not observed for the subsonic calculations. Also, there is no longer a region of negative

Figure 5.10: Contours of Constant Mach Number at Midspan (25% damaged, $i = 0^\circ$, $M_{te} = 1.14$).

Figure 5.11: Contours of Constant Mach Number at $Z/R_d \approx 4$ (25% damaged, $i = 0^\circ$, $M_{te} = 1.14$).

Figure 5.12: Pitchwise Mass-Averaged Total Pressure Loss for Two Transonic Mach Numbers ($i = 0^\circ$).

Figure 5.13: Pitchwise Mass-Averaged Trailing Edge Deviation for Two Transonic Mach Numbers ($i = 0^\circ$).

losses at the centre of the cutout, as was observed with the subsonic calculations. The peak trailing edge deviation can be seen to increase with trailing edge Mach number, although the spanwise extent of the under turned region remains about the same.

It is interesting to recall the findings in Section 4.1.4, where negative losses were observed at low Mach numbers. The fact that negative losses were observed for subsonic damaged blade calculations and not for transonic damaged blade calculations suggests that the distortion of the grid caused by modelling the blade damage may have increased the calculation's sensitivity to low Mach numbers. This may produce a localized region, near the cutout, where Mach number must be higher than usual to achieve realistic results.

Figure 5.14: Mass-Averaged Total Pressure Loss for Damaged and Undamaged Blade

Mach Number Effects on Mass-Averaged Results

Figure 5.14 shows mass-averaged total pressure loss as a function of Mach number, for both the undamaged and 25% damaged blades at design incidence (the other incidences have similar results). The losses rise significantly in the transonic range, as is expected due to the additional shock losses. The losses for the 25% damaged cascade do not rise as fast with M_{te} as the losses for the undamaged cascade. As a result, the losses for the 25% damaged cascade are quite a bit lower than for the undamaged cascade at $M_{te} = 1.14$.

It is clear that more investigation is required for using the Dawes code to calculate damaged cascade solutions near or in the transonic flow range. While the

transonic solutions calculated in the present research give some interesting results, no conclusions should be drawn from them until corresponding experimental results are obtained to verify the calculations.

Chapter 6

Conclusions and Recommendations

The objective of the present research was to investigate the effects of trailing edge damage on the loss characteristics and flow turning ability of a cascade of turbine blades. The measurements are to assist the development of correlations for the effects of damage on the blade aerodynamics. Such correlations will be used in gas turbine simulations related to engine health monitoring.

The major conclusions are as follows:

- (i) The effect of trailing edge damage on the loss characteristics of a turbine blade is small, at least under incompressible flow conditions. Therefore the efficiency of a turbine stage with trailing edge damaged rotor blades should remain essentially unchanged.
- (ii) A turbine blade with trailing edge damage produces considerable under turning of the flow in the area of the cutout. This implies that a turbine stage that has a rotor with trailing edge damaged blades will produce less power output for a given flow rate and rotor speed. Also, the under turning of the flow will have an impact on the performance of the following stages, since it will at least alter the incidence of the following blade row.

- (iii) The Dawes code has the potential to be a powerful tool for investigating the aerodynamics of damaged turbine blades. The Dawes code predicted many of the flow features observed in the experimental measurements. The loss peaks associated with the edges of the cutout, the significant under turning of the flow in the area of the cutout, and the small change in overall losses were all well predicted. The most notable discrepancy was the negative losses predicted at the centre of the cutout. The Dawes code proved its ability to calculate solutions in the transonic range, although no conclusions can be drawn from the predictions without supporting experimental data.

The following are recommended for future study:

- (i) Further low-speed experiments are required to investigate the aerodynamics of turbine blades with trailing edge damage. Measurements should be made for additional cutout sizes. Also, surface oil flow visualization should be performed for the blade surfaces of the damage blades, to assist in interpreting the measurements.
- (ii) The scope of the experimental measurements should be extended into the high subsonic and transonic flow regime. This would also provide data to compare with the high subsonic and transonic predictions calculated using the Dawes code.
- (iii) The use of the Dawes code to predict the aerodynamics of trailing edge damaged turbine blades must be investigated further. Notably, the reasons why negative losses are produced in the centre of the cutout need to be determined.

References

- [1] Abbott, M.A., *The Aerodynamic Effects of Roughness on the Performance of a Turbine Cascade*, Master of Engineering Thesis, Department of Mechanical and Aerospace Engineering, Carleton University, 1993.
- [2] Ainley, D.G., Mathieson, G.C.R., "A Method of Performance Estimation for Axial-Flow Turbines", Aeronautical Research Council of Great Britain, Reports and Memoranda No. 2974, 1951.
- [3] Aker, G.F., Saravanamuttoo, H.I.H., "Predicting Gas Turbine Performance Degradation due to Compressor Fouling Using Computer Simulation Techniques", ASME Paper 88-GT-206, 1988.
- [4] Cohen, H., Rogers, G.F.C., Saravanamuttoo, H.I.H., *Gas Turbine Theory*, 3rd Ed., Longman, 1987.
- [5] Danias, G., *Contributions to the Study of Tip Leakage Flow in a Planar Cascade of Turbine Blades*, Master of Engineering Thesis, Department of Mechanical and Aerospace Engineering, Carleton University, 1987.
- [6] Dawes, W.N., (a) *A Computer Program for the Analysis of Three Dimensional Viscous Compressible Flow in Turbomachinery Blade Rows*, Whittle Laboratory, Cambridge University, United Kingdom, 1986.
- [7] Dawes, W.N., (b) "Application of Full Navier-Stokes Solvers to Turbomachinery Flow Problems", von Karman Institute for Fluid Dynamics Lecture Series *Numeri-*

cal Techniques for Viscous Flow Calculations in Turbomachinery Bladings, January, 1986.

- [8] DeCook, S.J., King, P.I., Elrod, W.C., "Wake Mixing and Performance of a Compressor Cascade with Crenulated Trailing Edges", *AIAA Journal of Propulsion and Power*, Vol. 9, No. 2, March-April, 1993.
- [9] Goobie, S.M., *A Test Section for Investigating the Off-Design Performance of Planar Cascades*, Master of Engineering Thesis, Department of Mechanical and Aerospace Engineering, Carleton University, 1989.
- [10] Klein, W.A., *Contributions to the Study of the Aerodynamics of Damaged Turbine Blades*, Master of Engineering Thesis, Department of Mechanical and Aerospace Engineering, Carleton University, 1993.
- [11] Matteij, J.H.G., "Role of Brazing in Repair of Superalloy Components - Advantages and Limitations", *Materials Science and Technology*, August, 1985.
- [12] Muir, D.E., Rudnitski, D.M., Cue, R.W., "CF-18 Engine Performance Monitoring", *Engine Condition Monitoring - Technology and Experience*, AGARD-CP-448.
- [13] Rodger, P., *Establishing Two-Dimensional Flow in a Large-Scale Planar Turbine Cascade*, Master of Engineering Thesis, Department of Mechanical and Aerospace Engineering, Carleton University 1992.
- [14] Sjolander, S.A., Isaacs, D.P., Klein, W.A., "Aerodynamics of Turbine Blades with Trailing Edge Damage: Measurements and Computations", *ISOABE Eleventh International Symposium on Air Breathing Engines*, Vol. 2, pp. 1327-1334.
- [15] Tremblay, B., *Measurements and Correlation of Off-Design Performance of Axial Flow Turbines*, Master of Engineering Thesis, Department of Mechanical and Aerospace Engineering, Carleton University, 1989.

- [16] Turyk, P.J., "Compressor Fault Modelling for Engine Health Monitoring: Review and Recommendations", NRC Technical Report TR-ENG-004, NRC No. 31204, 1989.
- [17] Whitehouse, D.R., (a) *The Effect of Axial Velocity Ratio, Turbulence Intensity, Incidence and Leading Edge Geometry on the Off-Design Performance of a Turbine Blade Cascade*. Master of Engineering Thesis, Department of Mechanical and Aerospace Engineering, Carleton University, 1993.
- [18] Whitehouse, D.R., (b) "THP: Data Acquisition and Analysis Software", Department of Mechanical and Aerospace Engineering Report No. M&AE 93-1, Carleton University, 1993.
- [19] Yaras, M.I., *Flow in the Tip Gap of a Rectilinear Turbine Cascade*, Master of Engineering Thesis, Department of Mechanical and Aerospace Engineering, Carleton University, 1987.
- [20] Yaras, M.I., "Computational Schemes in Turbomachinery Flows", Department of Mechanical and Aeronautical Engineering Report, Carleton University, 1988.
- [21] Yaras, M.I., *Measurement and Prediction of Tip-Clearance Effects In a Linear Turbine Cascade*, Phd. Thesis, Department of Mechanical and Aerospace Engineering, Carleton University, 1990.
- [22] Yaras, M.I., Sjolander, S.A., Kind, R.J., "Effects of Simulated Rotation on Tip Leakage in a Planar Cascade of Turbine Blades", ASME Paper 91-GT-128.

Carleton University
Ottawa, Canada K1S 5B6

Text complete; leaf *105* omitted in numbering.

Appendix A

Additional Experimental Apparatus and Procedure Information

A.1 Data Acquisition Equipment List

Table A.1: Data Acquisition Equipment

Description	Make & Model	Serial Number	Comment
Pressure Transducer Channel #2	Rosemount 831A3	633	Calibration $1.382 Pa/mV$
Pressure Transducer Channel #4	Rosemount 831A3	591	Calibration $1.352 Pa/mV$
Pressure Transducer Channel #5	Baratron 220DC-00100A2BD -SPACL	79508-2	Calibration $0.686 Pa/mV$
Pressure Transducer Channel #6	Rosemount 831A3	567	Calibration $1.389 Pa/mV$
A/D Converter	Sciometric Instruments 81/902	9023J/I9006	
Pressure Switching Device	Scanivalve, 48 port		
Digital Voltmeter	DISA Type 55D31	9055F7211 325	

A.2 Windtunnel Checklist

The checklist shown in Figure A.1 enforces series of checks on the hardware and software settings, before the wind tunnel is started up. This allows the researcher to catch any settings that may have been improperly set, or perhaps forgotten altogether, before the test commences. There are also a series a checks to perform while the tunnel is operating, to make sure that there are no problems in the hardware/software data acquisition and control interface. The checklist includes a few blank lines, to allow checks that may be specific to a certain group of tests to be added. Since the wind tunnel operating times are recorded, the checklist makes an excellent log of wind tunnel operation time for wind tunnel maintenance purposes. It was found during the present research that, more than once, the checklist saved time consuming errors.

Appendix B

The Castle Preprocessor

The program Castle is designed to allow the user to construct a Dawes code input file modelling a linear cascade. As input, castle requires the (x, y) coordinates of the cascade blade profile. The user then specifies the following major parameters:

- The cascade stagger angle.
- The grid $I_{max} \times J_{max} \times K_{max}$ (i.e. pitchwise $\times$ streamwise $\times$ spanwise) dimensions.
- The degree of grid clustering in the pitchwise (I), streamwise (J), and spanwise (K) directions.
- The size and spanwise location of trailing edge damage (if any).
- The initial and boundary conditions used in the Dawes code calculations.

This document describes how to use the Castle preprocessor in preparing Dawes code input files. It is assumed that the reader is familiar with the Dawes code and its required input file format. For the reader unfamiliar with the Dawes code, it is suggested that Dawes (1986) ¹ be reviewed prior to using Castle.

¹Dawes, W.N., *A Computer Program for the Analysis of Three Dimensional Viscous Compressible Flow in Turbomachinery Blade Rows*, Whittle Laboratory, Cambridge University, United Kingdom, 1986.

To help the user become familiar with how Castle works, there is also an example Castle session, which can be found in Section B.7.

B.1 Overview of the Castle Program

B.1.1 Starting a Castle Session

Castle is started by simply typing `castle` from the UNIX command prompt. This assumes that the `castle` executable binary is either in the current directory or in the command `PATH`. The first thing Castle will do is look for the file `castle.defaults` in the current directory. If found, Castle will read all the default settings from the file and then give the `castle>` prompt. If `castle.defaults` is not found, an error is reported. More information concerning the *castle.defaults* file is contained in Section B.6.

B.1.2 Interacting with Castle

Castle is a command line driven program, command lines being entered at the `castle>` prompt. Each command line consists of three parts. The first word of the command line is the **command**, which informs Castle how to process the remainder of the command line. Examples of commands are **set**, **show**, and **plot**. Following the command there can be a number of *parameters keys*, which inform Castle exactly what parameter is involved in the command. Following the parameters key, if necessary, are the *settings*. These are words or numbers which Castle will apply to the parameter involved in the command. An example of a command line is:

```
castle> set spanwise clustering one-sided
```

where **set** is the command, **spanwise clustering** are the parameter keys, and **one-sided** is the setting.

To minimize the amount of typing required, Castle is able to recognize abbreviated commands. For example the command

```
castle> set trailing edge 20
```

can be abbreviated as

```
castle> set tra edg 20
```

The user can quickly establish recognized abbreviations through trail and error. Unrecognized commands will be rejected by Castle.

Throughout this manual, different commands and parameters will be introduced. To indicated the available settings to which a parameter can be set, a standard command line presentation will be used. For example:

```
castle> set pitchwise clustering {one-sided}{two-sided}{none}
```

means that the parameter pitchwise clustering has three possible settings, the words “one-sided”, “two-sided”, and “none”. Another Example:

```
castle> set blade rmax  $R_{max_b}$ 
```

means that the parameter blade rmax requires a numerical setting, the desired value of R_{max_b} .

B.1.3 The Set, Show, and Plot Commands

The **set**, **show**, and **plot** commands are the three essential commands used when interacting with Castle. Almost all of the directives entered into Castle by the user

will use the **set** command. A few examples are:

```
castle> set profile file cc3.profile
castle> set dimensions 33 107 33
castle> set damage trailing
```

A glossary of all the parameters accessed by the **set** command is contained in Section B.8.

Most parameters that are accessed using the **set** command have a corresponding **show** command. For example, given the **set dimensions** command above, we can use the **show dimensions** command as follows:

```
castle> show dimensions
the IxJxK dimensions are 33 107 33
```

The **plot** command is to be used after all the desired settings have been made. When the **plot** command is entered, Castle will create a Dawes code input file and save it in the output file specified by the **set output** command.

B.2 The Major Grid Sections and Directions

When specifying grid clustering, Castle handles each grid direction separately. In other words, specific clustering parameters exist and must be set for each grid direction. There are three major grid directions: the *streamwise* direction; the *pitchwise* direction; and the *spanwise* direction. These major directions are indicated in Figure B.1.

In order to simplify the specification of grid clustering in the streamwise directions, Castle divides the grid into three major sections: the *leading grid*; the *blade grid*; and

the *trailing grid*. The three sections are shown in Figure B.1. The leading grid is the portion of the grid upstream of the leading edge of the cascade. The blade grid is the portion of the grid between the leading and trailing edges of the cascade. The trailing grid is the portion of the grid downstream of the trailing edge of the cascade. When specifying the grid clustering in the streamwise direction, Castle handles each grid section separately.

Section B.3 contains further information, concerning grid clustering.

Figure B.1: The Grid Terminology used by Castle: (top) streamwise mesh; (bottom) meridional mesh.

B.3 Grid Clustering

Castle allows the user some control over the spacing of the grid nodes (i.e. the grid clustering) in the pitchwise, spanwise, and streamwise directions. In general, the user can specify two parameters: the clustering *type*; and the *maximum relative change in step size*. The clustering type controls whether the grid nodes will be clustered to one or both sides of the grid. The maximum relative change in step size, identified by the symbol R_{max} , controls how quickly the distance between grid nodes can vary while clustering.

Castle, in general, recognizes three clustering types: *one-sided*; *two-sided*; and *none*. For one-sided clustering, the grid nodes will be clustered towards only one side of the grid. The amount of clustering, and which side of the grid the nodes are clustered on, depends on the value of R_{max} . For two-sided grid clustering, the grid nodes will be clustered towards both sides of the grid, or towards the centre of the grid. The density of the grid clustering, and whether the the nodes cluster towards the sides or the centre, depends on the value of R_{max} . The higher the magnitude of R_{max} , the higher the density of grid clustering. $R_{max} = 1.0$ will produce no clustering, effectively the same as setting the clustering type to *none*.

Figure B.2 shows simplified examples of the three types of clustering.

B.3.1 Clustering in the Pitchwise and Spanwise Directions

The clustering in the pitchwise and spanwise directions requires the specification of the clustering type and R_{max} in each direction. For the pitchwise direction the clustering parameter commands are

```
castle> set pitchwise clustering {one-sided}{two-sided}{none}
castle> set pitchwise rmax  $R_{max_p}$ 
```

Figure B.2: Castle Clustering Types: (a) one-sided clustering; (b) two-sided clustering; (c) no clustering.

and for the spanwise direction the clustering parameters commands are

```
castle> set spanwise clustering {one-sided}{two-sided}{none}
castle> set spanwise rmax  $R_{max}$ .
```

The Dawes code recommends that both R_{max_p} and R_{max_s} be less than 1.30 and greater than 0.77.

Castle clusters the spanwise and pitchwise directions using a trigonometric algorithm, which varies the change in steps size from one grid point to the next with the maximum change in step size used equal to R_{max} :

$$R_n = \left[1 + \bar{R} \left| \cos \left(F \frac{n-1}{N-1} \right) \right| \right]^{\frac{|\cos(\pi \frac{n-1}{N-1})|}{\cos(\pi \frac{n-1}{N-1})}}, n = 1, N-1 \quad (\text{B.1})$$

where

R_n = the relative change in step size at the n 'th grid node

$\bar{R} = |R_{max}| - 1.0$

N = the number of points being clustered

$F = \pi$ for two-sided clustering

$F = \frac{\pi}{2}$ for one-sided clustering.

Castle will fit the N points to the required length (i.e. to the span of the blade for spanwise clustering, or to the blade pitch for pitchwise clustering), using the calculated values of R_n .

The value of R_{max} can be used to control the direction of clustering. For example, when the clustering type is two-sided, $R_{max} > 1.0$ will cluster the grid nodes to the edges of the grid. $R_{max} < 1.0$ will cluster the grid nodes to the center of the grid. When the clustering is one-sided, $R_{max} > 1.0$ will cluster the grid nodes to the $n = 1$ side, while $R_{max} < 1.0$ will cluster the grid nodes to the $n = N$ side. Note that, for one-sided clustering the algorithm of equation B.1 is not "symmetrical" about

$R_{max} = 1.0$. For example, a one-sided grid clustered with $R_{max} = 1.3$ is not a mirror image of a grid clustered with $R_{max} = 1/1.3 = 0.76923$. To produce a true mirrored grid, negative values of R_{max} can be used. When $R_{max} < 0$, Castle will cluster the grid as per equation B.1 and then mirror the results.

When blade damage is specified, the spanwise clustering parameters are no longer used. See Section B.4 for the clustering parameters to use for damaged blades.

B.3.2 Streamwise Clustering

Clustering in the streamwise direction is handled separately for each of the leading, blade, and trailing grid sections. This allows the grid to be clustered towards the leading and trailing edges of the blade while avoiding sudden changes in step size which will compromise the accuracy of the Dawes code calculations.

The clustering in the blade grid is handled in a similar manner as the clustering in the pitchwise and spanwise directions. The clustering parameters are:

```
castle> set blade clustering {one-sided}{two-sided}{none}
castle> set blade rmax  $R_{maxbl}$ 
```

Dawes code recommends that R_{maxbl} be less than 1.30 and greater than 0.77. For more information concerning the clustering algorithm used for the blade grid, see Section B.3.1.

The streamwise clustering for the leading and trailing grids uses a simple one-sided linear clustering algorithm, so there is no clustering type option. For the leading and trailing grids, the clustering parameters are then:

```
castle> set leading rmax  $R_{maxl}$ 
castle> set trailing rmax  $R_{maxt}$ 
```

respectively.

For the leading grid, the step size at the leading edge of the blade is set equal to the step size for the blade grid at the leading edge. That is,

$$X_{J_{le}} - X_{J_{le}-1} = X_{J_{le}+1} - X_{J_{le}} \quad (\text{B.2})$$

where X denotes axial coordinates and J_{le} denotes the index of the grid at the leading edge of the blade². The remainder of the leading grid nodes are spaced such that

$$X_n - X_{n-1} = R_{max_l}(X_{n+1} - X_n), n = J_{le} - 1, 1 \quad (\text{B.3})$$

For the trailing grid, the step size at the trailing edge of the blade is set equal to the step size for the blade grid at the trailing edge. That is,

$$X_{J_{te}+1} - X_{J_{te}} = X_{J_{te}} - X_{J_{te}-1} \quad (\text{B.4})$$

where X denotes axial coordinates and J_{te} denotes the index of the grid at the trailing edge of the blade². The remainder of the trailing grid nodes are spaced such that

$$X_{n+1} - X_n = R_{max_t}(X_n - X_{n-1}), n = J_{te} + 1, J_{max} \quad (\text{B.5})$$

While handling the streamwise grid clustering in this manner ensures that there will be no abrupt change in step size at the leading and trailing edges, a side effect is that the axial locations of the input and output planes become a function of R_{max_l} and R_{max_t} , respectively. The user should ensure that the resulting axial locations of the input and output planes are far enough from the blades, as recommended in Dawes (1986).

B.4 Adding Blade Damage

At present, Castle only supports the addition of trailing edge damage in the form of a semi-circular cutout in the blade's trailing edge. When including damage, the user can specify the diameter and the spanwise location of the cutout.

²The leading and trailing edge indices are set using the `set leading edge` and `set trailing edge` command, respectively. See the Glossary.

To inform Castle that damage is to be added to the blade, the user should use the **set damage** command

```
castle> set damage {trailing}{none}
```

Specifying *trailing*, informs Castle to add trailing edge damage, and specifying *none* informs Castle that no damage is to be added. By default, the damage parameter is always set to none. Note that there is a bug in Castle's programming that can lead to unpredictable results if the grid dimensions are changed after a **set damage trailing** command is entered. Therefore, if required, a **set dimensions** command should be entered before a **set damage** command.

B.4.1 Setting the Damage Size and Location

To set the size and location of trailing edge damage, the **set damage size** and **set damage location** commands are used:

```
castle> set damage size  $\frac{100D_d}{C}$   
castle> set damage location  $\frac{Y_d}{S}$ 
```

Figure B.3 illustrates how the damage size and location are specified. The damage size is entered as the damage diameter, D_d , expressed as a percentage of the blade chord, C . For example, the command **set damage size 25.0** will set the damage diameter equal to 25% of the blade chord. The damage location is specified as the spanwise location of the centre of the damage, normalized by the span of the blade. For example, the command **set damage location 0.5** will place the centre of the damage at midspan.

Figure B.3: Damage Size and Location Parameters

Figure B.4: The Three Damaged Grid Sections (indicated on meridional mesh with blade edges highlighted)

B.4.2 Spanwise Grid Clustering and Blade Damage

Adding damage to the trailing edge of the blade increases the complexity of the grid clustering in the spanwise direction. To accurately model the shape of the cutout there should be grid nodes that coincide with the top and the bottom edges of the cutout. To handle the spanwise grid clustering in the spanwise direction, Castle divides the spanwise direction into three sections, each with specific damage parameters: the *below-damage* section; the *above-damage* section; and the *in-damage* section. Figure B.4 shows the three damaged grid sections. The clustering type and R_{max} must be specified for each section. For the below-damage section, the clustering parameter commands are

```
castle> set damage clustering below {one-sided}{two-sided}{none}
castle> set damage rmax_below  $R_{max_b}$ 
```

For the in-damage section, the clustering parameter commands are


```
castle> set damage clustering in {one-sided}{two-sided}{none}  
castle> set damage rmax.in  $R_{max}$ ,
```

For the above-damage section, the clustering parameter commands are

```
castle> set damage clustering below {one-sided}{two-sided}{none}  
castle> set damage rmax_below  $R_{max}$ ,
```

The use of the clustering type and R_{max} parameters is discussed in detail in Section B.3. Note that it is possible to specify the clustering in the three damaged grid sections in such a way that abrupt changes in grid step size occur at the edges of the damage. When processing a grid (after a **plot** command is entered), Castle will check the relative change in step size in all three grid directions. If relative changes in step size greater than 1.3 or less than 0.77 are encountered, Castle will give a warning reporting the problem and the I , J , or K , index at where the problem occurred. It is up to the user to adjust the grid clustering to avoid such an occurrence.

B.5 The Profile Coordinates File

Castle requires that the blade profile (x,y) coordinate data be provided in a profile coordinates file which is opened and read when the **set profile file** command is entered. There are some formatting restrictions that need to be followed when creating the profile coordinate file for use with Castle:

Figure B.5: Specifying Blade Profile Coordinates

- (i) The (x,y) coordinates pairs should be specified one per line.
- (ii) The coordinate pairs should be specified starting from the leading edge, and progress in order in the counter-clockwise direction (see Figure B.5).
- (iii) The profile coordinate file must begin with header lines that contain a '#' in their first column. There can be any number of header lines, as long as the last two are used to specify the inlet and outlet metal angles, respectively. The FORMAT statement used to read these angles is (21X,F10.5).

Castle will locate the trailing edge of the blade by looking for the point farthest from the leading-edge point. Once the trailing-edge point is found, Castle will calculate the stagger angle of the blade in its given orientation. See Section B.9 for an example listing of a profile coordinate file.

B.6 The castle.defaults File

Due to the large number of options and parameters that must be set to fully specify a Dawes code input file, having to type in each option or setting would take a

prohibitively long time. The *castle.defaults* file is designed to pre-set all of the castle parameters to default values. If the desired grid is similar to the default grid, then the user should only have to change a few settings. The *castle.defaults* file must have all the parameters listed in the correct order and in the correct format, or else Castle will not be able to properly read the file. An example of the *castle.defaults* file is shown in Section B.9.

The *castle.defaults* file is read immediately after the Castle program is started. Castle expects to find *castle.defaults* in the current directory. If the file is not found, or if Castle cannot read the file for some reason, an error message will be displayed and the `castle>` prompt will be given. While it is possible to set all the required Castle parameters manually in the event that the defaults file cannot be read, this is not recommended.

Note that although damage parameters are included in the *castle.defaults* file, the design of the Castle program does not support the setting of blade damage through the defaults file. The damage type indicator should therefore always be set to *none*.

B.7 An Example Castle Session

The following example shows, step by step, how Castle can be used to generate a Dawes code input file. In general, most users will find Castle sessions are very similar to this example.

Starting the Castle session

Castle is started by typing `castle` from the UNIX command prompt. This assumes that the Castle executable binary is in the current directory or the command PATH. After reading in the information in the *castle.defaults* file, Castle will give the `castle>` prompt.

Specifying a profile description file

To specify to Castle the file containing the (x, y) coordinates describing the cascade blade profile, use the **set profile file** command. Let's assume the profile coordinates are stored in a file called *cc3.profile*. The command will then be

```
castle> set profile file cc3.profile
```

Setting the grid dimensions

The grid dimensions must be specified using the **set dimensions** command. For this example, the dimension will be $(I_{max} \times J_{max} \times K_{max}) = 33 \times 107 \times 33$.

```
castle> set dimensions 33 107 33
```

Setting the streamwise grid clustering

The streamwise grid clustering must be specified for each of the three major grid sections. For the leading and the trailing grid, only one-sided clustering is allowed. Therefore only the maximum relative change in step size, R_{max} , must be specified for these sections. For the blade grid, the clustering type as well as R_{max} , must be specified.

```
castle> set leading rmax 1.15
castle> set trailing rmax 1.15
castle> set blade clustering two-sided
castle> set blade rmax 1.3
```

Both R_{max_l} and R_{max_t} have been set to 1.15. For the blade grid, the clustering type has been set to two-sided, and R_{max_b} to 1.30 (the maximum recommended R_{max}).

Set the pitchwise and spanwise clustering

Both the clustering type and the maximum relative change in step size must be specified for the pitchwise and spanwise clustering.

```
castle> set pitchwise clustering two-sided
castle> set pitchwise rmax 1.25
castle> set spanwise clustering none
```

For this example, the pitchwise clustering is two-sided with $R_{max_{pitch}} = 1.25$, and there is no spanwise clustering. Pitchwise and spanwise clustering parameters need only be specified once for the entire grid (i.e. the settings apply for the leading, blade, and trailing grid sections).

Setting the output file name

To indicate what name to store the created Dawes code input file under, use the **set output** command. For this example, the created file will be named *dawes.in*

```
castle> set output dawes.in
```

If the **set output** command is not given, a default file name of *output.dat* will be used.

Creating and saving the Dawes code input file

To save the Dawes code input file that conforms to the above setting, use the **plot** command

```
castle> plot
```

Castle will generate the Dawes code input file and save it as *dawes.in*. While generating the file, Castle will check the relative change in step size in the streamwise, pitchwise, and spanwise directions, for every finite volume cell of the grid. If the maximum of 1.3 is exceeded, the (I, J, K) coordinate of the cell will be reported with a warning message.

B.8 Glossary of all Castle Parameters

This section contains a listing of all the **set** parameters recognized by Castle. The listing is broken down into two parts: Geometric Parameters, the parameters that determine the geometry of the grid; and Boundary, Flow Condition, and Numerical Parameters, the parameters that specify the boundary conditions, the fluid properties (i.e. specific heat ratio, Reynolds Number), and the numerical properties (i.e. artificial viscosity, turbulence model used).

Note that all of the **set** commands described have a corresponding **show** command, which will display the current value of the parameter.

B.8.1 Geometric Parameters

[set blade N_{blades}] Set the number of blades around the circumference of the hub.

The Dawes code uses this number to calculate the pitch of the blade row.

[set blade clustering {*one-sided*}{*two-sided*}{*none*}] Set the type of grid clustering for the blade grid.

[set blade rmax R_{maxbl}] Set the blade grid maximum relative change in step size, R_{maxbl} . The Dawes code recommends that the maximum relative change in step size should not exceed 1.3.

[set damage {*none*}{*trailing*}] Set the mode of damage for the cascade. *none* specifies an undamaged cascade, *trailing* specifies a semi-circular cutout at the trailing edge.

[set damage clustering above {*one-sided*}{*two-sided*}{*none*}] Set the type of spanwise grid clustering for the grid from the upper extent of the blade damage to the casing.

- [set damage clustering below {one-sided}{two-sided}{none}]** Set the type of spanwise grid clustering for the grid from the lower extent of the blade damage to the hub.
- [set damage clustering in {one-sided}{two-sided}{none}]** Set the type of grid clustering to be applied inside of the blade damage.
- [set damage location $\frac{Y_d}{S}$]** Set the location of the centre of the damage, Y_d , normalized by the span of the blade, S . For example: **set damage location 0.5** sets the centre of the damage at midspan.
- [set damage rmax_above R_{max_a}]** Set the maximum relative change in step size for the computational grid from the upper extent of the blade damage to the casing. R_{max_a} is applied to the spanwise clustering of this portion of the grid.
- [set damage rmax_below R_{max_b}]** Set the maximum relative change in step size for the computational grid from the lower extent of the blade damage to the hub. R_{max_b} is applied to the spanwise clustering of this portion of the grid.
- [set damage rmax_in R_{max_i}]** Set the maximum relative change in step size for the computational grid across the blade damage. R_{max_i} controls the spanwise clustering of this portion of the grid.
- [set damage size $\frac{100 \times D_d}{C}$]** Set the diameter of the damage, D_d , as a percentage of the blade chord, C . For example: **set damage size 25.0** sets the diameter of the damage to 25% of the blade chord. The blade chord is calculated automatically from the blade profile coordinates.
- [set dimensions $I_{max} J_{max} K_{max}$]** Set the $(I_{max} \times J_{max} \times K_{max})$ dimensions of the computational grid. For the Dawes code: $I \equiv$ pitchwise direction; $J \equiv$ streamwise direction; $K \equiv$ spanwise direction. The maximum allowable dimensions are $(I_{max} \times J_{max} \times K_{max}) = (37 \times 107 \times 37)$.

- [set leading edge J_{le}]** Set the J index of the leading edge of the cascade. J_{le} must be less than J_{te} which is set using the **set trailing edge** command.
- [set leading rmax R_{max_l}]** Set the leading grid's maximum relative change in step size, R_{max_l} , in the streamwise direction.
- [set pitchwise clustering {one-sided}{two-sided}{none}]** Set the type of grid clustering for the pitchwise direction.
- [set pitchwise rmax R_{max_p}]** Set the maximum relative change in step size, R_{max_p} , for the pitchwise direction
- [set profile file *filename*]** Specify the *filename* that contains the (x, y) profile coordinates of the cascade blade. When this command is entered, castle will immediately open *filename* and read the profile coordinate data. The profile data will be scaled by the factor set with the **set profile scaling** command. See the section on profile coordinate data for more information.
- [set profile scaling F_{PS}]** Set the profile scaling factor to F_{PS} . Every (x, y) coordinate read from the profile coordinates file will be scaled by this factor.
- [set spanwise clustering {one-sided}{two-sided}{none}]** Set the type of grid clustering for the spanwise direction.
- [set spanwise rmax R_{max_s}]** Set the maximum relative change in step size, R_{max_s} , for the spanwise direction
- [set stagger γ]** Set the stagger angle, γ , of the cascade, expressed in degrees. The Dawes code will rotate the input profile coordinates to the new stagger angle.
- [set time_steps N_{ts}]** Set the number of time steps, N_{ts} , for which the solution is to be calculated.

[set trailing edge J_{te}] Set the J index of the trailing edge of the cascade. J_{te} must be greater than J_{le} which is set using the set leading edge command.

[set trailing rn.ax R_{max_i}] Set the trailing grid's maximum relative change in step size, R_{max_i} . The Dawes code recommends that the maximum relative change in step size not exceed 1.3.

B.8.2 Boundary, Flow Condition, and Numerical Parameters

[set artificial_viscosity α_2 α_4 α_a] Set the artificial viscosity parameters for the Dawes code calculations.

[set cp c_p] Set the specific heat, c_p , of the gas flowing through the cascade. The specific heat is expected to have units of $\frac{J}{kgK}$.

[set dumps N_{ts1} N_{ts2} N_{ts3} N_{ts4} N_{ts5}] Set the time steps at which the Dawes code should dump its solution to a file. The five time steps should be in ascending order, with N_{ts5} equal the total number of time steps.

[set gamma γ] Set the specific heat ratio, γ .

[set k.thickness K F_{thick}] Set the thickness factor, F_{thick} , for the K 'th index. The thickness factors only need to be specified if tip clearance is desired (see set k_{tip}). This allows for the tip geometry to be "rounded" slightly. By default, the thickness factors are set to 1.0 for all K .

[set k.tip K_{tip}] Set the K index of the blade tip. For no tip clearance, K_{tip} should equal the maximum index, K_{max} . For tip clearance, K_{tip} must be set less than K_{max} . Also see set k.thickness.

[set multigrid artificial_viscosity α_{2h}] Set the artificial viscosity coefficient, α_{2h} , for the coarse grid.

[set multigrid {on}{off}] Specify whether to use multigrid calculations (*on*), or not to use multigrid calculations (*off*).

[set multigrid time_steps N_{ts_M}] Set the number of time steps, N_{ts_M} , for which to calculate the solution on the coarse grid only, before starting multigrid calculations.

[set multigrid time_step_multiplier F_{T_M}] Set the time step multiplier that is to be used for the coarse grid.

[set shear_stress_model *model#*] Set the shear stress model that the Dawes code is to use during the calculations. Valid *model#*'s are: 0 - for hub and casing shear stress from log law; 1 - for inviscid hub; 2 - for inviscid casing; 3 - for inviscid hub and casing; 4 - for hub and casing shear stress from laminar law. Note that the inviscid endwall models (available through *model#*'s 1, 2, and 3) is not implemented properly in the Dawes code.

[set time_step_multiplier F_T] Set the time step multiplier to be used for the calculations.

[set turbulent_length_scale EL_{max}] set the maximum turbulence length scale, EL_{max} , to be used for the calculation of the turbulence shear stress model.

[set rpm N] Set the rotational speed, N , of the rotor in rpm. Note that for cascades, N should equal 0 (i.e. set rpm 0).

[set inlet_hub_pressure $P_{up_{hub}}$] Set the absolute hub static pressure, $P_{up_{hub}}$, at the grid inlet, in Pa. This represents a first guess only.

[set inlet_meridional_velocity V_m] Set the meridional velocity, V_m , at the grid inlet, in meters per second.

[set inlet_pitch_angle β_r] Set the pitch angle, β_r , at the grid inlet, in degrees.

- [set inlet_tip_pressure $P_{up_{tip}}$]** Set the absolute tip (casing) static pressure, $P_{up_{tip}}$, at the grid inlet, in Pa . This represents a first guess only.
- [set inlet_stagnation_pressure $P_{o_{in}}$]** Set the absolute stagnation pressure, $P_{o_{in}}$, at the grid inlet, in Pa . This represents a fixed boundary condition.
- [set inlet_stagnation_temperature $T_{o_{in}}$]** Set the stagnation temperature, $T_{o_{in}}$, at the grid inlet, in degrees Kelvin. This represents a fixed boundary condition.
- [set inlet_swirl_angle β_s]** Set the absolute swirl angle, β , at the grid inlet, in degrees. When the rotational velocity, N , is zero, β equals the incidence angle.
- [set inlet_tangential_velocity V_t]** Set the tangential velocity, V_t , at the grid inlet, in meters per second.
- [set outlet_hub_pressure $P_{d_{hub}}$]** Set the absolute hub static pressure, $P_{d_{hub}}$, at the grid outlet, in Pa . This is a set boundary condition.
- [set outlet_tip_pressure $P_{d_{tip}}$]** Set the absolute hub static pressure, $P_{d_{tip}}$, at the grid outlet, in Pa . This is a fixed boundary condition.

B.9 Sample File Listings

A Sample *castle.defaults* File

```
stagger angle (deg)           : -21.6
incidence angle (deg)         : 3.0
deviation angle (deg)         : 0.0
number of blades               : 614
hub radius (mm)                : 1000.00
tip radius (mm)                : 1030.00
grid dimensions IxJxK         : 33 107 33
leading edge J                 : 20
trailing edge J                : 84
view                           : horizontal
leading grid clustering        : none
trailing grid clustering       : none
blade grid clustering          : two-sided
pitchwise grid clustering      : two-sided
spanwise grid clustering       : two-sided
display option                 : grid
output file                    : output.dat
output format                  : dawes
profile coordinates multiplying factor : 20.0
leading grid max relative change in step size : 1.15
trailing grid max relative change in step size : 1.15
blade grid max relative change in step size : 1.05
pitchwise grid max relative change in step size : 1.25
spanwise grid max relative change in step size : 1.0
maximum number of time steps   : 1500
wall shear stress model        : 0
multigrid indicator            : 1
coarse grid time steps         : 0
coarse grid time step multiplier : 2.0
coarse grid artificial viscosity : 0.0
specific heat, cp (J/kgK)      : 1005.4
specific heat ratio            : 1.4
fine grid time step multiplier : 0.5
Reynolds' number               : 300000.0
maximum turbulent length scale : 10.0
rounds per minute of blade rotor : 0.0
absolute inlet static pressure at hub (Pa) : 80000.0
absolute inlet static pressure at tip (Pa) : 80000.0
```

absolute outlet static pressure at hub (Pa)	: 71448.0
absolute outlet static pressure at tip (Pa)	: 71448.0
absolute inlet stagnation pressure (Pa)	: 100000.0
absolute inlet stagnation temperature (K)	: 300.0
absolute inlet tangential velocity (m/s)	: 160.0
inlet meridional velocity (m/2)	: 200.0
absolute inlet swirl angle (deg)	: 28.35
inlet meridional pitch angle (deg)	: 0.0
iteration dump indicators	: 500
	: 750
	: 1000
	: 1250
	: 1500
fine grid 2nd order artificial viscosity	: 1.0
fine grid 4th order artificial viscosity	: 0.01
fine grid artificial viscosity coefficient	: 2.00
damage type indicator	: none
damage size in percent of chord	: 25.0
damage edge radius ratio	: 2.0
damage location in percent of span	: 50.0
number of in-damage grid nodes	: 7
below damage spanwise clustering	: one-sided
below-damage max relative change in step size	: -1.25
in-damage spanwise clustering	: two-sided
in-damage maximum relative change in step size	: 1.3
above damage spanwise clustering	: one-sided
above damage max relative change in step size	: 1.25

A Sample Profile Coordinates File

```
# File: blade.cc3.profile
#
# inlet metal angle : 25.3500
# outlet metal angle: 57.5050
0.003252 0.273255
0.001177 0.285749
0.000039 0.293625
0.000482 0.301569
0.002488 0.309269
0.005980 0.316420
0.010817 0.322737
0.012576 0.324593
0.016093 0.328182
0.021368 0.333282
0.028404 0.339596
0.035435 0.345417
0.045986 0.353367
0.060053 0.362776
0.074120 0.371003
0.088187 0.378214
0.102255 0.384543
0.116322 0.390062
0.130389 0.394832
0.144456 0.398911
0.165557 0.403800
0.186658 0.407283
0.214793 0.409835
0.252927 0.410039
0.271062 0.407910
0.299196 0.403426
0.327331 0.396521
0.355465 0.387083
0.383600 0.374989
0.411734 0.360105
0.439869 0.342305
0.468003 0.321571
0.496138 0.297459
0.524272 0.270116
0.552407 0.239330
0.580542 0.205067
0.608676 0.167435
```

0.629777	0.137186
0.650878	0.105463
0.664945	0.083662
0.679012	0.061482
0.693080	0.039034
0.703630	0.022184
0.708905	0.013741
0.712422	0.008113
0.714181	0.005299
0.715259	0.002937
0.715693	0.000378
0.715453	-0.002207
0.714556	-0.004644
0.713062	-0.006767
0.711072	-0.008434
0.708720	-0.009533
0.706164	-0.009989
0.703577	-0.009772
0.701133	-0.008896
0.698997	-0.007421
0.697312	-0.005445
0.695676	-0.002433
0.692403	0.002510
0.687493	0.009743
0.677674	0.023584
0.664581	0.040835
0.651489	0.056839
0.638396	0.071729
0.618757	0.092230
0.599118	0.110775
0.572933	0.132889
0.546748	0.152478
0.520563	0.169822
0.494378	0.185146
0.468193	0.198665
0.442008	0.210541
0.415823	0.220882
0.389638	0.229790
0.363453	0.237357
0.337268	0.243664
0.311083	0.248791
0.284898	0.252823

0.258713	0.255849
0.232527	0.257960
0.206342	0.259260
0.186704	0.259775
0.167065	0.259958
0.153972	0.259917
0.140880	0.259756
0.127787	0.259484
0.114695	0.259109
0.101602	0.258641
0.088510	0.258082
0.075417	0.257437
0.065598	0.256898
0.059051	0.256515
0.052505	0.256111
0.047595	0.255797
0.044322	0.255581
0.042686	0.255471
0.034733	0.255729
0.026988	0.257556
0.019759	0.260880
0.013330	0.265570
0.007957	0.271439

Appendix C

Description of The Dawes Code Post-Processor, Centrepont

The Centrepont post-processor is designed to allow the user to extract data from results files produced by the Dawes 3-Dimensional Navier-Stokes Solver, in a form that is useful for further analysis. In addition to the data stored in a Dawes code results file, Centrepont can calculate the total pressure loss coefficients, Y , and C_{po} , the flow angle, and the trailing edge deviation. Centrepont has been designed with the analysis of cascade geometries in mind. Other geometries may present unexpected difficulties.

Centrepont minimizes the amount of interpolation that is necessary to extract data from a Dawes code results file by actually calculating the coordinates of the finite volume cell centres (where the Dawes code assumes the flow values exist). This method of extracting data has proven to be more effective than interpolating the data to the known coordinates of the cell corners.

This document describes how to use the Centrepont post-processor. It is assumed that the reader is familiar with the Dawes code and its conventions. For the reader unfamiliar with the Dawes code, it is suggested that Dawes (1986)¹ be reviewed prior

¹Dawes, W.N., *A Computer Program for the Analysis of Three Dimensional Viscous Compressible Flow in Turbomachinery Blade Rows*, Whittle Laboratory, Cambridge University, United

to using Centrepont.

C.1 Overview of the Centrepont Program

C.1.1 Starting a Centrepont Session

Centrepont is started by simply typing **centrepont** from the UNIX command prompt. This assumes that the centrepont executable file is either in your current directory, or in the command PATH. The first thing Centrepont will do is prompt the user for the grid data file, that is, the file in which the Dawes code has stored all of the grid coordinate data. Once the name of the grid data file has been entered, Centrepont will read in the grid coordinate data and then proceed to calculate the coordinates of the cell centres. This process can take many minutes, depending on the amount of computer and disk activity at the time. A counter is displayed to indicate how far along the process Centrepont is.

Once all the cell centre coordinates have been calculated, Centrepont will present the **Executive Menu**. This menu acts as a gateway to all the functions provided by Centrepont.

C.1.2 Interacting with Centrepont

Centrepont is primarily a menu driven program. Each menu presents a number of options or settings, each prefixed with an alpha-numerical character in brackets (**()**). To select an option or setting, simply type the corresponding number or letter (letter options are not case sensitive) and hit **<RETURN>**.

Most settings, when selected, prompt for the desired setting. If a setting is chosen accidentally, entering a period, **"."**, will usually abort the prompt without changing anything (though this feature is not universal).

C.2 The Executive Menu

The **Executive Menu** acts as a gateway to all the functions provided by Centrepont. Upon first entering the Executive Menu it should appear as follows:

```
-----  
-----> CENTREPOINT <-----  
-----  
-----Executive Menu-----  
-----
```

```
(G) Current Grid File      : nod120.grid  
(D) Current Results File  : -----  
(O) Current Output File   : -----
```

```
(1) Extract Grid Data.  
(2) Extract Flow Data.  
(Q) Quit CENTREPOINT.
```

Enter Choice:

The options are:

- (G) Current Grid File:** The current grid data file that is loaded into Centrepont. In this case, the file is *nod120.grid*.
- (D) Current Results File:** The current Dawes code results file that is loaded into Centrepont. In this case, no results file has yet been loaded.
- (O) Current Output File:** The current file where Centrepont output should be written to. In this case, no output file has yet been specified.
- (1) Extract Grid Data:** This option will take the user to the **Extract Grid Data Menu**, where different portions of the grid can be written to the output file as three dimensional surface data, which can be displayed using GNUPLOT V3.2 (or higher). **NOTE:** this menu is not yet implemented.

(2) Extract Flow Data: This option will take the user to the **Extract Flow Data Menu**, where different flow variables can be extracted from the Dawes code results file. These features are described in the next section.

(Q) Quit CENTREPOINT: This option will exit the user from the Centrepoint program.

C.3 The Extract Flow Data Menu

To enter the **Extract Flow Data Menu**, select option (2) from the Executive Menu. This should be done immediately after entering the Executive Menu for the first time (i.e. do not specify a Current Results File yet).

When entering the Extract Flow Data Menu for the first time, it should appear as follows:

```
-----
-----> CENTREPOINT <-----
-----
-----Extract Flow Data  Menu-----
-----

(1) Current Grid File      : nod120.grid
(2) Current Results File  : -----
(3) Current Output File   : -----
(4) Current Projection    : -----
(5) Endwall Interference  :   0 hub nodes,   0 case nodes
(6) Mass Averaging        : -----
(7) Location of X         : -----
(8) Normalized R          : -----
(9) Normalized R*THETA    : -----
(10) Extraction Variable  : -----
(11) Flow Vector Scaling  : -----
(12) Design Outlet Angle  : -----
(E) Return to Executive
(P) Process

Input plane located at X = -0.95055E-02
Output plane located at X =  0.31494E-01

Option:
-----
```

The following describes the various options and settings in detail.

(1) Current Grid File

This option allows the user to select a grid data file to be used in the data extraction process. When (1) is selected, the user will be prompted for the name of the new grid data file. Once the name is entered, the grid data will be read from the file and the coordinates of the cell centres will be calculated. A grid data file must be loaded before any data extraction can be performed.

It is up to the user to ensure that the grid coordinates described in the grid data file are appropriate for the specified Dawes code results file.

(2) Current Results File

This option allows the user to select a Dawes code results file from which data is to be extracted from. When (2) is selected, the user will be prompted for the name of the new Dawes code results file. Once the name is entered, the flow data will be read from the file. A Dawes code results file must be loaded before any data extraction can be performed.

It is up to the user to ensure that the specified Dawes code results file matches the specified grid data file.

(4) Current Output File

This option allows the user to specify the file to which all Centrepoint results are written when the **(P) Process** option is selected. When (4) is selected, the user is prompted for the name of the output file. Once entered, the previous output file is closed, and the new one is opened. If the selected file name already exists, Centrepoint will prompt whether are not to replace it.

Once an output file is opened, it will remain open until a new output file is specified or Centrepoint is exited. If option (P) is selected more than once while the same output file is specified, previously stored results will be overwritten. Note that since the output file is not closed until processing is completed, the new data will not

be available to other programs (like GNUPLOT) until the file is closed by specifying a new output file or quitting Centrepoin_t (this is because UNIX holds some data in a buffer until the file is closed).

(4) Current Projection

This option allows the user to select which projection to extract the data from. A better word than projection might be plane, since this option specified the orientation of the plane from which data is to be extracted. There are four different "projections" available: the *Horizontal* projection; the *Vertical* projection; the *Axial* projection; and the *Full* projection.

The *Horizontal* projection will extract data from a surface that has a constant normalized radial coordinate. If a horizontal projection is chosen, the user will be prompted for the radial coordinate. The entered coordinate should be normalized so that 0.0 refers to the surface coinciding with the hub, and 1.0 refers to the surface coinciding with the casing.

The *Vertical* projection will extract data from a surface that has a constant normalized $r\theta$ coordinate (i.e. the surface is essentially parallel to the blade surfaces). If a vertical projection is chosen, then the user is immediately prompted for the constant $r\theta$ coordinate. The entered coordinate should be normalized so that 0.0 refers to the surface that coincides with the suction surface of the blade, and 1.0 refers to the surface that coincides with the pressure surface of the blade.

The *Axial* projection will extract data from a surface that has a constant axial coordinate. If an axial projection is chosen, the user is immediately prompted for the constant axial coordinate. The axial coordinate entered must be between the input and output planes that are indicated at the bottom of the Extract Flow Data Menu.

The grid surfaces (i.e. the streamwise, meridional, and quasi-orthogonal surfaces that make up the computational mesh) will, in general, not match the extraction surface (i.e. the surface for which data is being extracted). Therefore, to obtain the

extraction surface data, Centrepont will interpolate between adjacent grid surfaces as required. The distribution of nodes on the extraction surface will therefore appear much like the distribution of node on the grid surfaces.

The *Full* projection will calculate the mass averaged value of the extraction variable for every quasi-orthogonal surface of the computational mesh. These values will be written to the output file along with the axial coordinates of the surfaces.

(5) Endwall Interference

This option allows the user to specify regions, near the endwalls of the blade passage, that will not be included in mass averaging calculations. This allows the effects of the endwall boundary layers to be excluded from mass averaging calculations

To specify endwall interference, the user must indicate how many grid nodes should be excluded from the mass averaging calculations. When (5) is chosen a list, showing what percentage of the blade span will be excluded for each number of grid nodes, is displayed. The number of nodes excluded does not have to be the same at the hub and the case.

(6) Mass Averaging

This option allows the user to turn mass averaging ON or OFF. Centrepont is able to calculate pitchwise mass averaged and overall mass averaged values of the selected extraction variable. This calculation is available only for axial projections.

When mass averaging is ON, both pitchwise mass averaged values and mass averaged values are calculated. The pitchwise mass averaged values are calculated according to

$$X' = \frac{\int_{0.0}^{1.0} X \rho V_x d(\frac{Y}{S})}{\int_{0.0}^{1.0} \rho V_x d(\frac{Y}{S})} \quad (C.1)$$

where

X = the extraction variable

V_x = the axial velocity

$\frac{Y}{S}$ = the spanwise coordinate normalized by the blade pitch.

The pitchwise mass averaging calculation is performed for every line of nodes along the blade span.

The mass averaged value is calculated according to

$$\bar{X} = \frac{\int_{0.0}^{1.0} \int_{0.0}^{1.0} X \rho V_x d(\frac{Y}{S}) d(\frac{Z}{H})}{\int_{0.0}^{1.0} \int_{0.0}^{1.0} \rho V_x d(\frac{Y}{S}) d(\frac{Z}{H})} \quad (C.2)$$

where

$\frac{Z}{H}$ = the spanwise coordinate normalized by the included height of the blade

The included height of the blade used in the mass averaging calculations is the height that lies between the endwall interference nodes. Therefore, all the data associated with the endwall interference nodes will be excluded from the calculation. The output file will contain the mass averaged value of the extraction variable, and the pitchwise mass averaged values as a function of the radial coordinate. The pitchwise mass averaged data can be plotted using the GNUPLOT plotting program.

As noted above, mass averaging is available only for axial projections. Turning mass averaging ON for other projections will lead to an error when the data extraction is processed.

When mass averaging is OFF, no mass averaging is performed. This is available for all projections. The extraction variable will be written to the output file as three-dimensional surface data, and can be displayed using the GNUPLOT plotting program.

(7) Location of x , (8) Normalized r , and (9) Normalized $r\theta$

These options allow the user to change the location of the extraction surface. Only one of these options is valid at any time, depending on the indicated extraction projection. Axial projections are associated with x coordinates, horizontal projections are associated with normalized r coordinates, and vertical projections are associated with normalized $r\theta$ coordinates.

When a projection is selected, the user is prompted for the associated coordinate so that the surface coordinate can be set. See the above section on (4) Current Projection for more details.

(10) Extraction Variable

This option allows the user to select what flow variable to extract from the surface extracted from the results file. When this option is selected, the user is presented with a menu of available extraction variables, which appears as follows:

Extract one of:

- (1) Axial Velocity
- (2) Tangential Velocity
- (3) Radial Velocity
- (4) Mach Number
- (5) Static Pressure
- (6) Static Density
- (7) Phi (whatever that is!)
- (8) Dynamic Viscosity
- (9) Total Pressure Loss Coefficient, C_{po}
- (10) Total Pressure Loss Coefficient, γ
- (11) Projected Flow Velocity Vectors
- (12) Horizontally Projected Flow Angle

Enter Choice:

Extraction variables (1) to (8) are available in the Dawes code solution file, and do not need to be derived. The two total pressure loss coefficients are calculated from

the available data as follows:

$$C_{po} = \frac{P_{o_{in}} - P_o}{q_{in}} \quad (C.3)$$

$$Y = \frac{P_{o_{in}} - P_o}{q_{exit}} \quad (C.4)$$

where q is the dynamic pressure and the subscripts *in* and *exit* denote the leading edge and trailing edge of the cascade, respectively. The dynamic pressure is calculated as the difference in the mass averaged values of the stagnation pressure and static pressure (calculated at the leading or trailing edge, whichever is relevant), and stagnation pressures are calculated using the isentropic compressible flow relation

$$\frac{P_o}{P_s} = \left(1 + \frac{\gamma - 1}{2} M^2\right)^{\frac{\gamma}{\gamma - 1}} \quad (C.5)$$

The variable Projected Flow Velocity Vectors is supposed to extract the magnitude and direction of the flow velocity vectors. Using the GNUPLOT plotting package, the flow velocity vectors could be displayed. Unfortunately, this feature does not work correctly and should not be used.

The variable Horizontally Projected Flow Angle calculates the angle of the flow velocity vectors projected on to the "horizontal" plane. This angle is calculated according to

$$\beta = \alpha_{offset} - \tan^{-1} \left(\frac{V_t}{V_x} \right) \quad (C.6)$$

In the FORTRAN source code, the $\tan^{-1}()$ function is calculated using the ATAN2 function, in order to handle all four quadrants. The α_{offset} is provided in case β is to be calculated relative to some angle not coincident with the axial direction. For example, to calculate the trailing edge deviation, set α_{offset} equal to the outlet metal angle of the blade. The angle α_{offset} is set using option (12) Horizontally Projected Angle Offset.

(11) Flow Vector Scaling

This option allows the user to set the scaling factor used to size the flow vectors calculated when Projected Flow Velocity Vectors is chosen as the extraction vari-

able. Unfortunately, the Projected Flow Velocity Vectors variable does not calculate reliably, so this option is of little use.

(12) Horizontally Projected Angle Offset

This option sets the reference angle to be used for calculating the extraction variable Horizontally Projected Flow Angle. See the (10) Extraction Variable entry above.

(P) Process

When this option is chosen, Centrepont will attempt to extract the current extraction variable from the current results file, following all the settings displayed in the Extract Menu. The results will be saved in the current output file (any existing data being overwritten). If a conflict occurs in any of the Extract Menu settings (for example, indicating mass averaging for a non-axial projection), Centrepont will abort the extraction processing.

When the extraction processing is complete, the Extract Menu will be redisplayed.

C.4 Centrepont Output Files

Centrepont output files are formatted for plotting with the GNUPLOT (version 3.2 and up) plotting program. All Centrepont output files begin with a similar header, which is essentially a copy of the Extract Menu. This allows the user to see exactly what the Extract Menu settings were when (P) Process was selected:

```
# -----
# Tue May 18 15:07:21 1993
# -----
# Current Grid File      : nod120.grid
# Current Results File   : nod121.out1.ts1250
# Current Output File    : nod121.beta.av.40%cdwn.ts1250
# Current Projection     : Axial
# Endwall Interference   : 6 hub nodes, 6 case nodes
# Mass Averaging         : On
# Location of X          : 0.02
# Normalized R           : -----
# Normalized R*THETA     : -----
# Extraction Variable    : Beta
# Flow Vector Scaling    : -----
# Input plane located at X = -0.10230E-01
# Output plane located at X = 0.32750E-01
# Maximum value: 0.13099E+03 at
# x: -0.75854E-02
# y: 0.10258E+01
# (i,j,k): 20 99 28
# Minimum value: 0.80260E+02 at
# x: -0.13802E-01
# y: 0.10023E+01
# (i,j,k): 8 99 3
# -----
# Fully Mass Averaged Value: -0.34821E+01
# -----
```

Note that there are some extra information which may be useful for analysis purposes. The date and time at the top indicates when the output file was created. The maximum and minimum values displayed indicates the maximum and minimum

values found for the last variable extracted from the current results file. As a consequence, when mass averaging is performed, the values will represent $\rho \times V_x$, and not the actual extraction variable. This is evident in the example shown above. The maximum and minimum values are accompanied by their locations in the flow field, expressed in (x, y) and (I, J, K) coordinates (I, J, K represent the pitchwise, streamwise, and spanwise grid node indices, respectively).

If mass averaging is performed, the mass averaged value of the extraction variable is indicated. This value excludes any nodes indicated by the Endwall Interference option.

After the file header there will be two or three columns of coordinate data, and a final column of extraction variable data. The exact number of columns depends on the projection chosen and whether mass averaging was performed. There are 5 different coordinate data columns:

(r) The radial coordinate. Used for axial and vertical projections.

(r_normalized) Normalized radial coordinate. Used for axial mass averaged projections. For this column the radial coordinates will be normalized so that 0.0 coincides with the hub endwall interference node and 1.0 coincides with the case endwall interference node.

(r*theta) The circumferential coordinate. Used for axial and horizontal projections.

(r*theta-offset) Offset circumferential coordinate. Used for horizontal projections. The offset subtracted from $r*theta$ is equal to the blade pitch. By using this column when plotting with GNUPLOT, the horizontal surface can be "doubled" to show the profile of the blade (see example below).

(x) The axial coordinate. Used for horizontal and vertical projections.

Using the *r*theta_offset* column with GNUPLOT

Since the use of the *r*theta_offset* column to display a “doubled” surface is probably not obvious, the following GNUPLOT example is included.

Use Centrepoint to produce a horizontal projection of Mach number, stored in the output file *mach.out*. To display a contour diagram, showing the profile of the blade, type **gnuplot** from the UNIX command prompt. Then type the following options.

```
gnuplot> set parametric
gnuplot> set view 0,0
gnuplot> set nosurface
gnuplot> set contour base
gnuplot> set cntrparam levels 20
```

These commands will set up GNUPLOT to display contour diagrams of the data in *mach.out*. To display the data, type

```
gnuplot> splot 'mach.out' using 1:2:4 with lines 0
gnuplot> replot 'mach.out' using 1:3:4 with lines 0
```

The **splot** command will display the plot the using the *r*theta* column as the “y-axis” data. The **replot** command will display the plot using the *r*theta_offset* column as the “y-axis” data. When the two plots are displayed side by side like this, the profile of the blade can be clearly seen, as shown in Figure C.1.

To display the mesh geometry, instead of the contour data, type

```
gnuplot> set nocontours
gnuplot> set surface
gnuplot> replot
```

Figure C.1: Example of GNUPLOT contour plot of Mach number.

END

3 1 - 0 5 - 9 5

FIN

Figure 5.1: Total Pressure Losses Measured in Damaged Blade Wakes:
 (a) 15% Cutout, $i = -10^\circ$; (b) 25% Cutout, $i = -10^\circ$;
 (c) 15% Cutout, $i = +10^\circ$; (d) 25% Cutout, $i = +10^\circ$.