INVESTIGATION OF THE EFFECTS OF AN UPSTREAM CAVITY ON THE SECONDARY FLOW IN TRANSONIC TURBINE BLADE PASSAGES WITH ENDWALL CONTOURING

by

Hamza M. Abo El Ella

B.Eng. - Aerospace

M.A.Sc. - Aerospace Engineering

A THESIS SUBMITTED TO THE FACULTY OF GRADUATE AND
POSTDOCTORAL AFFAIRS IN PARTIAL FULFILMENT OF THE
REQUIREMENTS FOR THE DEGREE OF

DOCTOR OF PHILOSOPHY

in

AEROSPACE ENGINEERING

Ottawa-Carleton Institute for
Mechanical and Aerospace Engineering

Department of Mechanical and Aerospace Engineering
Carleton University
Ottawa, Canada
November 2014

© 2014 Hamza M. Abo El Ella

Abstract

The investigation documented here aims to contribute to the understanding of secondary flows in modern high-pressure (HP) turbine blade passages. More specifically, it aims to improve the understanding of vortical structures near the endwall with respect to the presence of an upstream cavity that approximates the gap present in actual engines between the rotor and stator of an HP turbine. Further, it aims to assess the viability of using non-axisymmetric endwall contouring to reduce endwall losses, including those generated by the presence of an upstream cavity, using a modern airfoil, at HP turbine representative speeds and at off-design Mach numbers.

To attempt to achieve this, a combination of flow measurement, flow visualization, and computational fluid dynamics (CFD) with linear turbine cascades was used. The test matrix consisted of three cases all with one common blade cascade geometry (SL2P). SL2P was combined with one of three endwalls: a baseline flat endwall, a flat endwall with a cavity incorporated upstream of the blade row, and a contoured endwall with the same upstream cavity geometry. A combination of pressure probes, pressure taps, and flow visualization were used to collect quantitative and qualitative data in a blow-down type wind tunnel. Complementary CFD studies were also carried out using the commercial CFD code ANSYS CFX.

It was found that the presence of an upstream cavity can noticeably alter the structure and the strength of the secondary flow. When compared to the baseline flat endwall, measurements downstream of the trailing edge have shown a substantial increase in the size and strength of the passage vortex. The presence of the cavity also introduces a significant increase in the level of overturning. The secondary kinetic energy increased significantly due to the presence of the cavity, and a major effect on the overall losses was also evident, with the cavity endwall generating up to 14% higher mixed-out row losses

relative to the baseline flat endwall at the design Mach number of 0.80. It was also found that the endwall contouring design used can successfully lower the losses by altering the structure and the strength of the secondary flows. When compared to the cavity endwall, the contoured endwall results have shown a significant reduction in the size and strength of the counter vortex, accompanied with a large reduction in the secondary kinetic energy in the vortex-vortex interaction regions between oppositely rotating vortices. Overall, the contoured endwall showed a drop of up to 10.7% in mixed-out row losses relative to the cavity endwall, with slightly higher losses than the baseline flat endwall. In assessing the off-design performance of the endwall contouring, the experimental results showed that the endwall contouring can continue to successfully lower the losses at small off-design Mach numbers (Mach 0.69, 0.75, 0.84, and 0.89).

The computations did not agree well with the experiment, and overestimated both the losses and the secondary kinetic energy. They predicted almost equal losses for both the flat and cavity endwalls, and no benefit for the endwall contouring design used in this study.

“No one shall truly believe, until he loves for his brother that which he loves for himself”

- The Prophet Mohammed (PBUH) {Bukhari, 1:2:13}

Dedicated to the freedom and justice loving people of this world

Acknowledgements

I would like to acknowledge and thank my supervisor Dr. Steen A. Sjölander for his continuous support and guidance in completing this work. Special thanks to the professional staff of the Department of Mechanical and Aerospace Engineering for their technical assistance and contributions over the years.

The technical expertise of Dr. Thomas Praisner and the financial contributions of Pratt & Whitney USA are also acknowledged.

My deepest gratitude and dearest thanks to my mother and father, to whom I am eternally indebted for their many sacrifices and their support in all my endeavours. To my sister and brothers, my heart felt appreciation for always being there for me.

Contents

Abstract	ii
Acknowledgements	v
Table of Contents	vi
List of Figures	xii
List of Tables	xviii
Nomenclature	xix
1 Introduction	1
1.1 Research Motivation	1
1.2 Research Objectives and Methods	5
1.3 Organization of the Thesis	7
2 Background and Literature Review	8
2.1 Introduction	8
2.2 Turbine Cascade Flow Field and Losses	8
2.3 Engine Turbine Flow Field	12

2.4	Investigations on Secondary Flows in the Compressible Regime	13
2.5	Investigations on Control of Secondary Flows	16
2.5.1	Overview of Investigations	16
2.5.2	Endwall Fencing Investigations	19
2.5.3	Three Dimensional Blade Investigations	20
2.5.4	Leading Edge Modification Investigations	21
2.5.5	Axisymmetric Endwall Contouring Investigations	21
2.5.6	Non-Axisymmetric Endwall Contouring Investigations	22
2.5.7	Non-Axisymmetric Endwall Contouring Used in Current Investigation	24
2.5.8	Mixed Methods Investigations	26
2.6	Investigations on Upstream Cavity Effects	27
2.7	Previous High Speed Investigations at Carleton University	31
2.8	Summary and Conclusions	31
3	Experimental Method and Data Reduction	33
3.1	Introduction	33
3.2	High Speed Wind Tunnel	33
3.2.1	Overview of the Wind Tunnel	33
3.2.2	Wind Tunnel Test Section	35
3.3	Wind Tunnel Measurements and Instrumentation	37
3.3.1	Measurement Locations and Procedures	37
3.3.2	Seven-Hole Pressure Probe	40
3.3.3	Pressure Transducers	42
3.3.4	Control and Data Acquisition Systems	44
3.4	High Speed Probe Calibration Rig	45
3.4.1	Preliminary Design	45
3.4.2	Final Design	46

3.4.3	Control and Data Acquisition Systems	48
3.5	Test Cascade and Builds	50
3.5.1	Blade Geometry	50
3.5.2	Flat Endwall	51
3.5.3	Flat Endwall with Upstream Cavity	52
3.5.4	Contoured Endwall with Upstream Cavity	53
3.6	Flow Visualization Technique	54
3.6.1	Method and Equipment	54
3.6.2	Image Post-Processing	56
3.7	Data Reduction Methods and Uncertainty	57
3.7.1	Primitive Flow Variables	57
3.7.2	Flow Averaging	57
3.7.3	Losses	59
3.7.4	Mixed-out Loss	61
3.7.5	Loss-Breakdown Scheme	62
3.7.6	Exit Flow Angle Deviation, Streamwise Vorticity, and Secondary Kinetic Energy	62
3.7.7	Uncertainties in Measured Values	64
4	Computational Method	69
4.1	Introduction	69
4.2	Simulation Cases and Computational Domain	69
4.3	Grid Refinement and Solution Convergence	72
5	Flat and Cavity Endwalls at Design Mach Number	77
5.1	Introduction	77
5.2	Inlet Endwall Boundary Layers	78

5.3	Midspan Blade Loading Distributions	79
5.4	Inner-passage Flow Field Behaviour	80
5.4.1	Endwall Flow Visualization	80
5.4.2	Suction Surface Flow Visualization	87
5.4.3	Computational Flow Physics and Visualization	88
5.5	Downstream Flow Field Behaviour	92
5.5.1	Loss Distributions	92
5.5.2	Streamwise Vorticity Field	95
5.5.3	Secondary Kinetic Energy Field	98
5.5.4	Computational Downstream Results	101
5.6	Total and Mixed-out Losses	108
5.7	Discussion and Conclusions	110
6	Endwall Contouring Performance at Design Mach Number	113
6.1	Introduction	113
6.2	Inlet Endwall Boundary Layer	114
6.3	Midspan Blade Loading Distributions	114
6.4	Inner-passage Flowfield Behaviour	115
6.4.1	Endwall Flow Visualization	115
6.4.2	Suction Surface Flow Visualization	118
6.4.3	Computational Flow Physics and Visualization	119
6.5	Downstream Flow Field Behaviour	121
6.5.1	Loss Distributions	121
6.5.2	Streamwise Vorticity Field	122
6.5.3	Secondary Kinetic Energy Field	125
6.5.4	Computational Downstream Results	128
6.6	Total and Mixed-out Losses	133

6.7	Discussion and Conclusions	135
7	Off-Design Mach Number Behaviour	138
7.1	Introduction	138
7.2	Results for Below Design Mach Number	139
7.2.1	Inlet Endwall Boundary Layer	139
7.2.2	Midspan Blade Loading Distributions	140
7.2.3	Downstream Loss Distributions	140
7.2.4	Downstream Streamwise Vorticity Field	142
7.2.5	Downstream Secondary Kinetic Energy Field	144
7.3	Results for Above Design Mach Number	145
7.3.1	Inlet Endwall Boundary Layer	145
7.3.2	Midspan Blade Loading Distributions	146
7.3.3	Downstream Loss Distributions	147
7.3.4	Downstream Streamwise Vorticity Field	149
7.3.5	Downstream Secondary Kinetic Energy Field	150
7.4	Total Row Losses, and Loss Breakdown	152
7.5	Discussion and Conclusions	154
8	Conclusions, Contributions, and Recommendations	156
8.1	Conclusions	156
8.2	Contributions	158
8.3	Recommendations	159
	List of References	161
	Appendices	
A	Seven-Hole Probe Calibration and Data Reduction	178

B Pressure Transducers Calibration	181
C Flow Visualization Video Results	183
D Mixed-out Loss Calculations	185

List of Figures

1.1	Engine Alliance GP7000 – typical turbofan engine components.	2
1.2	Typical engine configuration showing cavity geometry.	5
2.1	The main secondary flow structures in a turbine blade passage.	11
2.2	Control points for generating endwall geometry and sample results.	25
3.1	Pratt & Whitney Canada High Speed Wind Tunnel.	34
3.2	Wind tunnel test section.	36
3.3	Experiment measurement locations.	38
3.4	Typical inlet uniformity, cavity endwall measurements shown.	39
3.5	Typical wake periodicity at 50% span.	40
3.6	Seven-hole pressure probe design and dimensions.	42
3.7	High speed probe calibration rig.	47
3.8	High speed probe calibration rig – motor control and data acquisition software user interface.	49
3.9	SL2P – blade geometry and parameters.	51
3.10	Flat endwall with upstream cavity.	52
3.11	Contoured endwall with upstream cavity.	53
3.12	Flow visualization setup and equipment.	55

3.13	Image post-processing done through colour channel manipulation.	57
4.1	Computational domain and boundary conditions.	70
4.2	Inlet total pressure (measured values) and computed turbulence profiles. .	71
4.3	Typical hexahedral grid generated.	73
4.4	Typical convergence history.	74
4.5	Typical comparison of computational and experimental midspan blade load- ing.	75
4.6	Comparison of CFD grids to experiment in terms of stagnation pressure loss coefficient at 1.4 C_x	75
5.1	Inlet endwall boundary layer comparison of baseline flat endwall and cavity endwall.	79
5.2	Experimental and CFD (PWA design prediction) blade loading comparison for flat and cavity endwalls.	80
5.3	Flat endwall flow visualization results, with interpreted separation lines on the right.	81
5.4	Cavity endwall flow visualization results with interpretation.	83
5.5	Cross-sectional views of the cavity vortex, at planes normal to the cavity. .	85
5.6	Computational results showing the formation of the cavity vortex.	85
5.7	Comparison of flat and cavity endwalls; suction surface flow visualization with interpretation.	87
5.8	Computational flow visualization for the flat endwall.	89
5.9	Computational flow visualization for the cavity endwall.	90
5.10	Comparison of baseline flat endwall and cavity endwall: (a) Downstream local stagnation pressure losses. (b) Pitchwise averaged stagnation pressure losses.	92

5.11	Comparison of baseline flat endwall and cavity endwall: (a) Streamwise vorticity field with stagnation pressure loss contour lines. (b) Pitchwise averaged exit flow angle deviation.	96
5.12	Comparison of baseline flat endwall and cavity endwall: (a) Secondary kinetic energy field with stagnation pressure loss contour lines and secondary velocity vectors. (b) Pitchwise averaged exit flow angle deviation.	99
5.13	Comparison of flat endwall and cavity endwall: (a) Secondary kinetic energy field with stagnation pressure loss contour lines and secondary flow pseudo-streamlines. (b) Pitchwise averaged secondary kinetic energy.	100
5.14	Comparison of downstream local stagnation pressure losses: (a) Experimental results. (b) Computational results.	102
5.15	Comparison of streamwise vorticity field with stagnation pressure loss contour lines: (a) Experimental results. (b) Computational results.	104
5.16	Comparison of secondary kinetic energy field with stagnation pressure loss contour lines and secondary flow pseudo-streamlines: (a) Experimental results. (b) Computational results.	106
5.17	Comparison of pitchwise averaged results: (a) Pitchwise averaged stagnation pressure losses. (b) Pitchwise averaged exit flow angle deviation. (c) Pitchwise averaged secondary kinetic energy.	107
5.18	Experimental and computational losses, and secondary kinetic energy.	108
6.1	Inlet endwall boundary layer comparison of flat, cavity, and contoured endwalls.	114
6.2	Experimental and CFD (PWA design prediction) blade loading comparison for flat, cavity, and contoured endwalls.	115
6.3	Comparison of cavity, and contoured endwalls; endwall flow visualization with interpretation.	117

6.4	Comparison of cavity, and contoured endwalls; suction surface flow visualization with interpretation.	119
6.5	Computational results showing Mach number distributions with velocity vectors and contouring features, at 0.61 mm above the endwall surface . .	120
6.6	Comparison of baseline flat endwall, cavity endwall, and contoured endwall: (a) Downstream local stagnation pressure losses. (b) Pitchwise averaged stagnation pressure losses.	121
6.7	Comparison of baseline flat endwall, cavity endwall, and contoured endwall: (a) Streamwise vorticity field with stagnation pressure loss contour lines. (b) Pitchwise averaged exit flow angle deviation.	123
6.8	Comparison of baseline flat endwall, cavity endwall, and contoured endwall: (a) Secondary kinetic energy field with stagnation pressure loss contour lines and secondary velocity vectors. (b) Pitchwise averaged exit flow angle deviation.	126
6.9	Comparison of baseline flat endwall, cavity endwall, and contoured endwall: (a) Secondary kinetic energy field with stagnation pressure loss contour lines and secondary flow pseudo-streamlines. (b) Pitchwise averaged secondary kinetic energy.	127
6.10	Comparison of downstream local stagnation pressure losses: (a) Experimental results. (b) Computational results.	128
6.11	Comparison of streamwise vorticity field with stagnation pressure loss contour lines: (a) Experimental results. (b) Computational results.	129
6.12	Comparison of secondary kinetic energy field with stagnation pressure loss contour lines and secondary flow pseudo-streamlines: (a) Experimental results. (b) Computational results.	131

6.13	Comparison of pitchwise averaged results: (a) Pitchwise averaged stagnation pressure losses. (b) Pitchwise averaged exit flow angle deviation. (c) Pitchwise averaged secondary kinetic energy.	133
6.14	Experimental and CFD total losses, and secondary kinetic energy.	134
7.1	Inlet endwall boundary layer comparison of flat, cavity, and contoured endwalls, below design Mach number.	139
7.2	Experimental and CFD (PWA design prediction) blade loading comparison for flat, cavity, and contoured endwalls below design Mach number.	140
7.3	Downstream local stagnation pressure losses, below design Mach number: (a) Flat endwall. (b) Cavity endwall. (c) Contoured endwall.	141
7.4	Streamwise vorticity field with stagnation pressure loss contour lines, below design Mach number: (a) Flat endwall. (b) Cavity endwall. (c) Contoured endwall.	143
7.5	Secondary kinetic energy field with stagnation pressure loss contour lines, below design Mach number: (a) Flat endwall. (b) Cavity endwall. (c) Contoured endwall.	145
7.6	Inlet endwall boundary layer comparison of flat, cavity, and contoured endwalls, above design Mach number.	146
7.7	Experimental and CFD (PWA design prediction) blade loading comparison for flat, cavity, and contoured endwalls above design Mach number.	147
7.8	Downstream local stagnation pressure losses, above design Mach number: (a) Flat endwall. (b) Cavity endwall. (c) Contoured endwall.	148
7.9	Streamwise vorticity field with stagnation pressure loss contour lines, above design Mach number: (a) Flat endwall. (b) Cavity endwall. (c) Contoured endwall.	150

7.10	Secondary kinetic energy field with stagnation pressure loss contour lines, above design Mach number: (a) Flat endwall. (b) Cavity endwall. (c) Contoured endwall.	151
7.11	Total losses and loss break-down; decreasing mach number (black to blue), and increasing Mach number (black to red).	152
7.12	Variation of mixed-out losses with Mach number.	153
A.1	Mach 0.77 seven-hole probe calibration surfaces: C_α (top left), C_β (top right), C_o (bottom left), and C_q (bottom right).	179
B.1	Pressure transducers calibration data.	182
C.1	Video image captures during different phases of wind tunnel operation. . .	183
C.2	Suction surface flow visualization showing false separation.	184
D.1	Mixed-out loss calculation - control volume and variables.	186

List of Tables

2.1	Summary of investigations on secondary flows in the compressible regime. .	14
2.2	Summary of investigations on control of secondary flows (1960 – 2002). . .	17
2.3	Summary of investigations on control of secondary flows (2002 – 2014). . .	18
2.4	Summary of investigations on upstream cavity effects.	29
3.1	Pressure transducers.	43
3.2	Uncertainty values.	67
4.1	Summary of grid quality	74
B.1	Pressure transducers calibration coefficients.	182

Nomenclature

Symbols

A	Area.
a	Speed of sound.
C	Blade true chord length.
C_x	Blade axial chord length.
C_{ω_s}	Streamwise vorticity coefficient.
C_{P_o}	Total pressure coefficient.
C_{SKE}	Secondary kinetic energy coefficient.
F_x	Force in the x-direction.
F_y	Force in the y-direction.
F_z	Force in the z-direction.
H	Boundary layer shape factor, $H = \frac{\delta^*}{\theta}$.
h	Blade span.
i	Incidence.
k	Turbulence kinetic energy.
M	Mach number.
m	Transducer calibration curve slope.
P	Static pressure.

P_o	Stagnation pressure.
q	Dynamic pressure.
R	Universal gas constant.
Re	Reynolds number.
s	Blade pitch, or entropy.
T	Static temperature.
T_o	Stagnation temperature.
Tu	Turbulence level.
u	Axial component of velocity.
V	Flow velocity, or transducer voltage.
v	Pitchwise component of velocity.
w	Spanwise component of velocity.
x	Axial coordinate.
Y	Stagnation pressure loss coefficient.
y	Pitchwise coordinate.
z	Spanwise coordinate, or output voltage at zero pressure.
$\dot{E}$	Energy rate.
$\dot{m}$	Mass flow rate.

Greek Symbols

α	Spanwise angle.
β	Pitchwise angle.
β_1	Inlet flow angle.
β_2	Outlet flow angle.
γ	Heat capacity ratio.
δ	Uncertainty.

δ^*	Boundary layer displacement thickness.
$\delta_{99\%}$	Boundary layer thickness 99%.
ζ	Stagger angle.
θ	Boundary layer momentum thickness.
ρ	Density.
σ	Standard deviation.
ϕ	Flow variable.
ψ	Exit flow angle deviation.
ω	Vorticity, or specific turbulence dissipation rate.

Subscripts

1	Cascade inlet.
2	Cascade outlet.
CL	Centre line value.
is	Isentropic condition.
mix	Mixed-out plane value.
ms	Midspan value.
sec	Secondary, i.e. secondary velocity components.

Superscripts

"	Planewise mass-averaged value.
'	Pitchwise mass-averaged value.
**	Planewise area-averaged value.
*	Pitchwise area-averaged value.

Abbreviations

3D	Three-Dimensional.
AC	Annular Cascade.
AEC	Axisymmetric Endwall Contouring.
CCD	Charge-Coupled Device.
CFD	Computational Fluid Dynamics.
Cont.	Contoured.
D/S	Downstream.
EF	Endwall Fencing.
Fluor.	Fluorescent.
HP	High-Pressure.
HSAL	High Speed Aerodynamics Laboratory.
LC	Linear Cascade.
LEM	Leading Edge Modifications.
LP	Low-Pressure.
NEC	Non-axisymmetric Endwall Contouring.
NIST	National Institute of Standards and Technology.
PID	Proportional Integral Derivative.
PWA	Pratt & Whitney Aircraft.
PWC	Pratt & Whitney Canada.
RANS	Reynolds-Averaged Navier-Stokes.
RTR	Rotating Test Rig.
SST	Shear-Stress-Transport.
TDB	Three-Dimensional Blades.
TE	Trailing Edge.
Vis.	Visualization.

Introduction

1.1 Research Motivation

The gas turbine, first successfully introduced to aircraft propulsion in the form of the turbojet engine in the 1930s, has evolved over time to become one of the most efficient and reliable means of aircraft propulsion today (Saravanamuttoo *et al.*, 2009). Tomorrow's engines strive to attain these same or better levels of efficiency and reliability with a lighter and cheaper overall design. Gas turbine designers are constantly striving for improved efficiency while maintaining reliability of the engines. While in recent years there have been significant increases in overall thermal efficiencies in industrial applications of gas turbines, there have been limited gains in the already highly efficient gas turbines for aircraft propulsion. Industrial applications have the advantage of being able to recover large amounts of the waste heat for co-generation or combined cycle use. They are also not limited by size and weight constraints, and thus facilitate the use of large intercooling and regenerative thermal cycles. On the other hand, aircraft engines are limited by both size and weight constraints, which has led to the development of compressors and turbines with very high isentropic efficiencies and with compact combustor designs of similarly high combustion efficiencies.

Figure 1.1: Engine Alliance GP7000 – typical turbofan engine components.

The Engine Alliance GP7000 shown in Figure 1.1, is a modern turbofan aircraft engine currently in service on the Airbus A380. The figure shows the typical components of a turbofan engine, which include: the fan, the low-pressure (LP) compressor, the high-pressure (HP) compressor, the combustor, the LP turbine, and the HP turbine. Researchers and engineers are constantly seeking ways to increase the performance by optimizing each of these components. Higher efficiencies than current levels, however, are very hard to attain, with increases in performance reaching diminishing returns; this has led designers to shift their focus to reducing the weight and cost of a gas turbine while maintaining current

levels of efficiency and reliability. Advancements in materials sciences have allowed for lighter components; for example, the fan component of the GP7000 features hollow titanium blades (Kandebo, 2004). Advanced light materials are also used downstream of the fan in the LP, and HP compressors, and recently to a more modest degree, in the LP, and HP turbine.

From an aerodynamics design perspective, one of the ways to reduce the weight of the engine is by increasing the blade loading—that is the lift force contributed by each blade—to lower the number of blades, leading to “high-lift” airfoil designs. This results in an overall lighter and cheaper engine, ultimately yielding a more economical and fuel efficient aircraft. Turbine blades are typically more expensive than compressor blades (this is due to the higher manufacturing and materials costs associated with the higher thermal stress requirements and complex internal structures needed for cooling). Thus, engine designers seeking a reduction in both weight and cost, have targeted the turbine component of the gas turbine engine for the introduction of high-lift airfoils. The LP turbine typically making up approximately 20%, and up to a third of the overall engine weight (Curtis *et al.*, 1997), is an obvious target, and has been the focus of past and ongoing research. High-lift aerodynamics are currently key characteristics of modern LP turbines, including the LP turbine of the GP7000 shown above (Weber & Hackenberg, 2007).

Increasing the blade loading, however, is not without disadvantages. The increased loading is typically accompanied with increased secondary flow losses. These losses are due to three dimensional flow effects (flows that are “secondary” to the main stream path, also referred to as endwall flows) near the endwalls of the turbine blades, and negatively impact efficiency. To mitigate these secondary flow losses, researchers have been actively pursuing ways to control the secondary flows in such a way as to reduce losses. One of the successful developments from this research has been the use of endwall contouring, a

combination of hills and valleys at the endwall, optimized to control the secondary flows in such a way as to reduce losses. The LP turbine of the Rolls-Royce Trent 900 engine, also currently powering Airbus A380 aircraft, incorporates endwall contouring in its design (Gonzalez *et al.*, 2006).

The research undertaken here focuses on the secondary flow in the HP turbine, as highlighted in Figure 1.1. HP turbines unlike the LP turbines, bear the full brunt of the hot high pressure combustion gases, expanding through the HP turbine stages at significantly higher Mach numbers than that of the LP turbine. Therefore, from a design perspective, HP turbines present a considerable challenge due to the high aerodynamic, thermal, and mechanical loads present. In addition, since HP turbines have considerably lower aspect ratios than LP turbines, the effects of secondary flow losses are more significant than on LP turbines. With engine manufacturers beginning to look to implement high-lift technology in HP turbines as well as LP turbines, there is a strong motivation to understand ways of controlling the secondary flows at the high speed conditions representative of HP turbines.

In actual engine turbine configurations (both LP and HP), a space exists between the rotor disk and the stator ring to allow for the required mechanical clearance for the rotor disks to rotate. This space manifests itself as a cavity at the rotor-stator interface as shown in Figure 1.2, where for the HP turbine cold flow bled from the compressor, known as purge flow, is ejected to cool the rotor disk, and prevent ingestion of the hot combustion gases into the disk space. In the case where the manipulation of endwall flow structures is of interest, there is a strong need to understand the effects of such an upstream cavity, and longer term, the effect of the purge flow, that could potentially have significant impact on endwall flows. The effects of platform geometry and purge flow have been investigated at low speeds before, however, very little research has been done at high speeds, i.e. compressible flow conditions, representative of HP turbines.

Figure 1.2: Typical engine configuration showing cavity geometry.

For these above motivations, the research undertaken involves transonic high lift blade cascade builds, tested at high speeds, with endwall contouring, that include an upstream cavity on their endwall plate. This upstream cavity is representative of the cavity between the rotor-stator in a turbine stage in actual engines, with the absence of purge flow. The research undertaken, its objectives and tasks, are described in the following section.

1.2 Research Objectives and Methods

The research undertaken is of both academic and practical interest. The research aims to contribute academically to the understanding of secondary flows in turbine blade passages.

More specifically, to improve the understanding of vortical structures near the endwall with respect to upstream cavity geometry—including control of their behaviour to reduce losses—in the context of highly loaded blades and at transonic speeds. The research aims to contribute practically by providing an assessment of the feasibility of a novel Pratt & Whitney Aircraft (PWA) high-lift airfoil, designed to reduce the secondary losses with somewhat higher blade loadings than currently used in HP turbines. Further, the research will reduce one of the gaps between cascade testing and testing under engine conditions by providing valuable high speed data and insights into the flow physics from a cascade with an integrated upstream cavity.

The author’s research aims to improve the understanding of the flow physics through a combination of linear cascade flow measurement, flow visualization, and computational fluid dynamics (CFD). Linear cascade testing is a useful first step for investigating and developing new turbine design concepts. While the flow is idealized compared with engine conditions (e.g. absence of radial pressure gradients), the cascade provides a controlled environment under which important flow effects can be isolated and investigated in detail. For example, low-speed cascade testing has already been used at Carleton University (MacIsaac, 2011) to investigate the aerodynamic effects of an upstream cavity, with and without purge flow, in a related turbine cascade. The present work adds the important influence of compressibility by extending the investigations to a transonic linear cascade (without purge flow). The next step in the development of the technology would be investigations in a rotating test rig. While such a rig gives a closer approximation to engine conditions, it generally provides only overall performance data, such as stage efficiency. Thus, rig tests would only be undertaken once computational studies and cascade testing, such as the ones presented in this thesis, have provided the physical understanding and have shown promising results.

The test matrix consists of three cases all with one common blade cascade geometry. This blade geometry, designated as SL2P, was developed using CFD by PWA to reduce the secondary flow losses by means of modifying the geometry of the airfoil. SL2P is combined with one of three endwalls to form the three test cases. The first endwall is a flat endwall and forms the baseline case, and will be referred to simply as the *flat endwall*. The second endwall is a flat endwall with a cavity incorporated upstream of the blade row, and is referred to as the *cavity endwall*. Lastly, the third endwall is a contoured endwall with the same upstream cavity geometry, and is referred to simply as the *contoured endwall*; the endwall contouring is of the non-axisymmetric type and is designed with the aim to reduce secondary flow losses. Experimental work was carried out at the Carleton University High Speed Aerodynamics Laboratory (HSAL) using the Pratt & Whitney Canada (PWC) High Speed Wind Tunnel. A combination of pressure probes, and pressure taps, combined with flow visualization was used to collect quantitative and qualitative data. Complementary CFD studies were carried out using ANSYS CFX.

1.3 Organization of the Thesis

The following chapter provides relevant background information, a review of the pertinent research in the open literature, as well as a brief summary of previous transonic studies at Carleton University. In Chapter 3, a description of the PWC High Speed Wind Tunnel, experiment equipment, test cascade, and experiment method is given. Chapter 4 provides a description of the computational method used. Detailed discussion of the results is given in Chapters 5 to 7. Finally, conclusions drawn from the discussed results are given in Chapter 8 along with recommendations for future work.

Background and Literature Review

2.1 Introduction

In this chapter, a general description of the turbine cascade flow field and losses is presented with a focus on secondary flows, followed by a brief comparison to the actual engine flow field. Next, a summary of the pertinent literature is presented with regards to the following: secondary flow physics and losses in the compressible flow regime, control of secondary flows, and upstream cavity geometry effects. In each section, the relevant literature is presented in chronological order in tabulated form, and the most pertinent literature from the table is then discussed. Finally, a brief overview of relevant research at Carleton University is also provided.

2.2 Turbine Cascade Flow Field and Losses

A linear turbine cascade depicts a stationary turbine blade row and endwall with a very large (essentially infinite) radius. While the cascade flow field is idealized, it provides a close approximation of the flow physics in an actual engine (see next section for comparison to engine flow field).

The general turbine cascade flow field is depicted in Figure 2.1. In a turbine passage, in addition to frictional effects in the boundary layer of the rotor blades, there are several other sources that contribute to the total losses in the passage. The losses due to each of these sources are normally estimated individually, and then simply added to estimate the total losses through the blade passage. Kacker & Okapuu (1982) provide a well established example of this. In axial turbomachinery, these losses can be subdivided into four components defined as:

Profile Losses:	These are losses generated by friction in the boundary layer formed around the surface of the blade. These include losses caused by rapid expansion at the trailing edge, and by the mixing out of the wake downstream of the blade.
Annulus Losses:	These are losses generated outside the blade passage by friction on the endwalls upstream and downstream of the passage.
Tip-Leakage Losses:	These are losses generated by flow escaping at the tip clearance of rotor blades. The losses are due to both viscous effects as well as due to mixing with the surrounding flow. The present research was carried out in the absence of tip clearance, and therefore tip-leakage losses are not within its scope.
Shock Losses:	These are losses due to the presence of shockwaves. In transonic turbines, shockwaves primarily appear at the trailing edge, but can appear locally inside the blade passage at regions of high curvature where high local velocities occur.
Endwall Losses:	These are losses generated inside the blade passage by the secondary three dimensional flow structures that generate additional losses through high shear stresses at the endwalls and by mixing with the downstream flow. These losses are the focus of this research and are further described below.

Some of the earliest investigations into the secondary flow field date back to the mid 70s, (Sjolander, 1975). In the following years, Sieverding (1985), Gregory-Smith (1997), and Langston (2001) have provided substantial reviews of the subject. Figure 2.1 shows the freestream flow entering the passage at the inlet, where it is then turned by the blade profile at the midspan region. As the flow enters the blade passage the cross-passage pressured gradient required to produce the midspan flow turning is largely imposed on the lower momentum flow near the endwalls leading to the formation of secondary flows (flows perpendicular to the main flow direction). Essentially, as the boundary layer is turned, the lower momentum fluid is accelerated more strongly towards the low pressure side of the blade passage than the higher momentum fluid closer to midspan. This results in a cross flow across the passage from the pressure side to the suction side of the passage. Then, in order to preserve continuity a vortical flow is formed; this is the *passage vortex*.

Additionally, due to the adverse pressure gradient as the flow approaches the leading edge of the blade, the boundary layer fluid separates at a saddle point, forming the “horse-shoe” vortex. This horse-shoe vortex is made up of a clockwise rotating pressure side leg, and a counter-clockwise rotating suction side leg. The pressure side leg of the vortex is drawn into the passage and swept towards the suction side of the blade by the cross-passage pressure gradient, and has the same sense of rotation as the passage vortex, and thus combines with it. As the passage vortex travels through the passage and towards the suction side of the blade, it can interact with the suction surface boundary layer giving rise to a suction surface *corner vortex*.

The counter-clockwise rotating suction side leg of the horse-shoe vortex is drawn into an adjacent passage, and develops into the *counter vortex*, an opposite sense of rotation to the passage vortex. The blade passage flow is a highly complex flow, and has been studied extensively; a detailed and widely accepted depiction of the flow field is provided by Wang *et al.* (1997).

These vortical structures (along with the induced endwall boundary layer) are responsible for the generation of endwall losses through two methods: viscous shearing between each other, and by viscous shearing and mixing with the freestream fluid. From this depiction of the passage flow, it is evident that by modifying the cross-passage pressure gradients, it may be possible to control the development of these vortical structures and thus attempt to limit their strength and reduce the losses generated by them. This will be discussed in more detail in Section 2.5.

Figure 2.1: The main secondary flow structures in a turbine blade passage.

2.3 Engine Turbine Flow Field

Linear cascade testing is an important first step in investigating and developing new turbine design concepts. It provides a controlled environment for the detailed investigation of important flow effects, but there are differences between its flow field and that of an actual turbine. The flow through an actual turbine blade row typically differs from that of a linear turbine cascade in the following respects:

Radial Pressure Gradients:	Actual turbines have an annular end-wall geometry resulting in radial pressure gradients.
Downstream Blade Row Effect:	In an actual engine, the presence of the downstream blade row alters the flow field in the preceding blade row.
Skewed Boundary Layer:	The inlet boundary layer of real machines is usually skewed by the change in relative velocity from a stationary stator row to a rotating rotor row.
Flow Unsteadiness:	Blade wakes from proceeding blade rows interact with subsequent blade rows resulting in an inherently unsteady flow.
Cooling Flows:	In most engine turbines, cooling flows are employed to cool both the blades and the endwalls.
Tip-Leakage Flows:	In unshrouded turbines, tip leakage flows can have significant impact on the flow field.
Combustor Exit Non-Uniformities:	Hot gas flow exiting the combustor typically have areas of varying temperatures that could result in non-uniform velocities at the inlet of the downstream blade row.

Additionally, in actual engines, platform cavity geometry is present upstream of the blades (see Figure 1.2), and many modern turbines operate in the transonic region, and may operate at off-design Mach numbers. In this research, however, the latter effects are partially captured by including an upstream cavity on the endwall, and testing at transonic conditions at design and off-design Mach numbers.

2.4 Investigations on Secondary Flows in the Compressible Regime

There is extensive literature on experimental investigations on secondary flows. Some investigations have focused on the effects of varying flow parameters on the secondary flow, and these will be addressed in this section. Other investigations have focused on the control of secondary flows to reduce losses, these will be addressed in the following section.

A substantial amount of the research has been carried out at low speeds. These investigations have varied several flow parameters to study their effects on the secondary flow physics, and the secondary flow losses. Benner (2003) provides an excellent and thorough review of these low-speed investigations. Similarly, in the compressible regime, several studies exist that have examined the effects of several flow parameters on the secondary flow, including incidence, Reynolds number, Mach number, inlet boundary layer thickness, flow turning, and loading distribution. These investigations are summarized in Table 2.1, where details of each investigation are presented under five main column headings: Author(s), Type of Study, Measurement Details, Parameters Investigated, and Flow Conditions. Within the Measurement Details column, the location and the type of measurements taken are identified. The abbreviations used in the table are listed directly under the table. The main observations and conclusions obtained from these studies will be briefly discussed.

Author(s)	Type of Study	Measurement Details						Parameters Investigated					Flow Conditions				
		Within Passage	D/S of TE	Blade Surface	Pressure Probe	Non Intrusive	Flow Vis.	i	Re	M	IBLT	FT	LD	i (°)	Re x 10 ⁻⁵	M ₂	Tu (%)
<i>Sieverding & Wipulte (1981)</i>	LC	✓	✓		✓					✓				design	2.6, 11.0, 18.0	0.1, 0.6, 0.8	0.5, 1.0, 1.0
<i>Binder & Romey (1983)</i>	RTR		✓		✓									design	2.6	0.74	-
<i>Camus et al. (1984)</i>	AC		✓	✓	✓		oil	✓						-10, 0, 5	3.6-10.6	0.7-1.2	-
<i>Hodson & Dorniny (1987)</i>	LC	✓	✓	✓	✓		oil							design	2.9	0.7	0.5
<i>Moharak et al. (1988)</i>	LC		✓		✓					✓				design	2.5-5.5	0.2-0.5	2-3
<i>Bassi et al. (1989)</i>	LC/CFD		✓							✓				design	-	0.5, 1.02, 1.38	-
<i>Perichizzi (1990)</i>	LC		✓	✓	✓					✓				design	-	0.2-1.5	-
<i>Moustapha et al. (1993)</i>	LC	✓		✓	✓		oil/schlieren			✓				design	2.0 (design)	0.3-1.36	0.2 (design)
<i>Perichizzi & Dossena (1993)</i>	LC		✓	✓	✓		oil	✓						-60 to 35	8.4	0.7	0.9
<i>Weiss & Fothner (1995)</i>	LC		✓	✓	✓		oil						✓	design	5.0	0.59	4.0
<i>Hermanson & Thole (2000)</i>	LC/CFD	✓				✓				✓				design	-	0.09-0.85	-
<i>Dossena et al. (2004)</i>	LC		✓	✓	✓					✓				-24, 0, 24	0.85 (design)	0.3-1.0	1.0
<i>Taremi et al. (2010)</i>	LC		✓	✓	✓		oil					✓		design	5.0	0.8	4.0
<i>Yasa et al.(2010)</i>	AC/CFD		✓		✓					✓				design	-	0.6, 0.95	-
<i>Abraham et al (2011)</i>	LC/CFD		✓	✓	✓					✓			✓	-10, 0, 10	-	0.6-1.1	8.0

Abbreviations:

LC: Linear Cascade
RTR: Rotating Test Rig
AC: Annular Cascade
CFD: Computational Fluid Dynamics
D/S: Downstream

TE: Trailing Edge
Vis.: Visualization
i: Incidence
Re: Reynolds Number
M: Mach Number

IBLT: Inlet Boundary Layer Thickness
FT: Flow Turning
LD: Loading Distribution
M₂: Outlet Mach Number
Tu: Turbulence

Table 2.1: Summary of investigations on secondary flows in the compressible regime.

Several investigations have examined the effect of off-design incidence on secondary losses. Camus *et al.* (1984) examined the effects of incidence with varying Mach numbers in an annular cascade, while Perdichizzi & Dossena (1993) measured secondary losses at off-design incidence for three linear cascades with different pitch-to-chord ratios. More recently, Dossena *et al.* (2004) examined five different linear cascades with different pitch-to-chord ratios and stagger angles at design and off-design conditions, and Abraham *et al.* (2011) examined three airfoil geometries in a linear cascade at off-design conditions. These investigations conclude that at positive incidence the secondary flow losses are generally higher, but they are not effected much by negative incidence. The investigators attribute the change at positive incidence to the increase in blade loading near the leading edge that yields stronger secondary flows resulting in higher endwall losses.

Sieverding & Wilputte (1981), Bassi *et al.* (1989), Perdichizzi (1990), Moustapha *et al.* (1993), and Yasa *et al.* (2010), all conducted investigations to examine the effects of Mach number on the secondary flows. The most notable conclusion was that for low to subsonic Mach numbers in the compressible regime only small changes in secondary flow losses are observed. In general, as the Mach number is increased, the secondary flow structures, in particular the passage vortex, move closer to the endwall along with peak loss regions associated with them. At higher Mach numbers in the transonic regime, secondary flow losses can decrease significantly, however, different loss coefficients, may exhibit contradictory trends in losses as the Mach number is raised.

Experimental studies examining the effects of inlet boundary layer thickness in the compressible regime are very limited. Mobarak *et al.* (1988) carried out measurements on linear cascades using three different geometries, and suggests an increase in inlet boundary layer thickness increases the secondary flow losses. Weiss & Fottner (1995) investigated the influence of loading distribution, by comparing two cascades, a front-loaded and an aft-loaded design. They found that the front-loaded design produced significantly higher

secondary losses than the aft-loaded design. They attributed the higher secondary losses to the strong cross-passage pressure gradient imposed by the suction peak near the leading edge (of the front loaded cascade), on the undisturbed boundary layer entering the passage. More recently Taremi *et al.* (2010a,b) presented results from four cascades: two low turning cascades with the same flow turning but different loading levels, and two high turning cascades with the same flow turning but different loading levels. They found that the higher loading in the low turning case resulted in higher secondary flow losses. In the high turning case, the cascade with the higher loading displayed stronger secondary flows. However, overall losses were not higher than the lower loaded cascade with the authors attributing this to a possible suction-side separation bubble on the higher loaded cascade.

2.5 Investigations on Control of Secondary Flows

2.5.1 Overview of Investigations

Pressure losses due to secondary flows in a typical turbine passage can be responsible for as much as 30% of the total losses in conventionally loaded airfoils, and even more for highly loaded airfoils, as demonstrated by Haselbach *et al.* (2002). Perhaps one of the easiest ways to reduce the secondary flow losses is by having a good two dimensional profile design (e.g. aft-loaded designs as discussed above) before three dimensional flow effects are considered. More elaborate methods for control of secondary flows involve three-dimensional modifications to the blade geometry, endwall geometry, or a combination of both. These methods can be classified as follows: *Endwall Fencing*, *Three-Dimensional Blades* (e.g. *lean*, or *sweep*), *Leading Edge Modifications* (e.g. *fillets*, or *bulbs*), *Axisymmetric Endwall Contouring*, and *Non-axisymmetric Endwall Contouring*.

A substantial number of investigations using these five methods are available in the literature. Tables 2.2 and 2.3 provide a summary of these investigations in chronological

Author(s)	Type of Study	Measurement Details						Secondary Flow Control				Flow Conditions			
		Within Passage	D/S of TE	Blade Surface	Pressure Probe	Non Intrusive	Flow Vis.	Method				i (°)	Re x 10 ⁻⁵	M ₂	Tu (%)
								EF	TDB	LEM	AEC				
Delich et al. (1960)	LC/AC	✓	✓	✓	✓				✓		-	-	-	-	-
Morris & Hoare (1975)	LC	✓	✓	✓	✓				✓		design	2	low subsonic	-	-
Kopper et al (1981)	LC	✓	✓	✓	✓		oil		✓		design	17, 19	0.85	1.8	-
Boyle & Hess (1982)	AC/LC/CFD		✓	✓	✓				✓		design	varied	varied	-	-
Boletis (1985)	AC	✓	✓	✓	✓				✓		design	2.5	-	0.8	-
Moustapha & Williamson (1986)	AC		✓	✓	✓		oil		✓		design	-	0.67 - 1.18	1.7	-
Atkins (1987)	LC	✓	✓	✓	✓				✓		design	-	-	-	-
Harrison (1990)	LC	✓	✓	✓	✓		oil-dot	✓			design	-	0.14	-	-
Chung et al (1991)	LC	✓	✓			✓	ink-dot				design	2.05, 2.93	-	-	-
Chung & Simon (1993)	LC		✓		✓		ink-dot/tuft	✓			design	2.14, 2.05	-	10, 0.6	-
Rose (1994)	CFD									✓	design	-	-	-	-
Harland et al (1998)	LC/CFD	✓	✓		✓					✓	design	0.6 (at probe)	-	5.1	-
Dossena et al (1999)	LC/CFD		✓	✓	✓					✓	design	10.8	0.60	0.5	-
Duden et al (1999)	LC/CFD		✓	✓	✓				✓ +		design	-	0.60	3.3	-
Aunapu et al (2000)	LC	✓	✓			✓	ink-dot/tuft	✓			design	1.8	-	8.0	-
Harvey et al (2000)	CFD									✓	design	-	-	-	-
Harland et al (2000)	LC/CFD	✓	✓	✓	✓		oil			✓	design	4	0.1	5	-
Gregory-Smith et al (2001)	LC	✓	✓	✓	✓		oil			✓	design	-	low subsonic	5	-
Brennan et al (2001)	CFD									✓	design	-	-	-	-
Rose et al (2001)	RTR		✓	✓	✓					✓	-	4.4 (rotor)	off/design	-	-
Sauer et al (2001)	LC/CFD	✓	✓		✓			✓			design	4.5	0.2	< 1	-
Harvey et al (2002)	RTR/CFD		✓	✓	✓					✓	design	4.4(HP), 1.76(IP)	design	-	-
Hoeger et al (2002)	LC		✓	✓	✓		oil				design	-	0.81-0.9 (inlet)	1.38 to 1.27	-
Zess & Thole (2002)	LC/CFD	✓		✓	✓			✓		✓	design	2.3	-	0.6	-

Abbreviations:

LC: Linear Cascade
RTR: Rotating Test Rig
AC: Annular Cascade
CFD: Computational Fluid Dynamics
D/S of TE: Downstream of Trailing Edge

Vis.: Visualization
EF: Endwall Fencing
TDB: Three-Dimensional Blades
LEM: Leading Edge Modifications
AEC: Axisymmetric Endwall Contouring

NEC: Non-axisymmetric Endwall Contouring
i: Incidence
Re: Reynolds Number
M₂: Outlet Mach Number
Tu: Turbulence

Table 2.2: Summary of investigations on control of secondary flows (1960 – 2002).

Author(s)	Type of Study	Measurement Details						Secondary Flow Control Method				Flow Conditions			
		Within Passage	D/S of TE	Blade Surface	Pressure Probe	Non Intrusive	Flow Vis.	EF	TDB	Method		i (°)	Re x 10 ⁵	M ₂	Tu (%)
										LEM	AEC				
Han et al (2002)	AC/LC		✓		✓		ink-dot	✓				design	8.3	0.3	-
Eymann et al (2002)	RTR		✓			✓		✓	✓			design	2.0	0.6	2 (at vane 1)
Gier et al (2002)	CFD							✓	✓			design	-	-	-
Becz et al (2004)	LC		✓	✓	✓					✓		design	5.9	-	-
Pasinato et al (2004)	LC/CFD	✓	✓		✓	✓				✓		design	-	-	-
Ingram et al (2005)	LC/CFD	✓	✓	✓	✓		oil				✓	design	-	-	5
Nagel & Baier (2005)	LC/CFD	✓	✓	✓	✓		oil	✓	-		✓	design	50	0.59	-
Bagshaw et al (2005)	LC		✓	✓	✓		oil	✓				design	-	-	4
Saha & Acharya (2006)	CFD											design	2.1	-	5
Sonoda et al (2007)	RTR/CFD		✓		✓					✓		design	35	1.04	-
Dorfner et al (2007)	CFD							✓			✓	design	-	-	-
Gustafson et al (2007)	LC	✓	✓	✓	✓		smoke				✓	design	2.3	-	4
Praisner et al (2007)	CFD/LC		✓	✓	✓						✓	design	1.65	-	-
Bagshaw et al (2008a)	CFD							✓			✓	design	4	-	-
Bagshaw et al (2008b)	LC		✓	✓	✓		paraffin	✓			✓	design	4	-	-
Gregory-Smith et al (2008)	LC		✓	✓	✓		oil/paraffin	✓	✓		✓	design	4	-	-
Schuepbach et al (2009)	CFT		✓		✓					✓		design	7.1/3.8/5.1	0.54/0.26/0.46	-
Snedden et al (2009)	RTR/CFD		✓		✓							design	1.3	-	< 1
Germain et al (2010)	RTR/CFD		✓	✓							✓	design	7.1/3.8/5.1	0.54/0.26/0.46	-
Knezevici et al (2010)	LC	✓	✓	✓	✓		oil				✓	design	1.26	0.111	4
Panchal et al (2011)	CFD											design	-	-	5
Taremi et al (2011)	LC		✓	✓	✓		oil				✓	design	6	0.78	4
Torre et al (2011)	LC/CFD		✓		✓		oil				✓	design	2.32	0.07	0.5
D'Ippolito et al (2011)	LC/AC/CFD	✓	✓	✓	✓		oil	✓				design	10.5(AC), 8.5(LC)	0.65	-

Abbreviations:

LC: Linear Cascade
RTR: Rotating Test Rig
AC: Annular Cascade
CFD: Computational Fluid Dynamics
D/S of TE: Downstream of Trailing Edge

Vis.: Visualization
EF: Endwall Fencing
TDB: Three-Dimensional Blades
LEM: Leading Edge Modifications
AEC: Axisymmetric Endwall Contouring

NEC: Non-axisymmetric Endwall Contouring
i: Incidence
Re: Reynolds Number
M₂: Outlet Mach Number
Tu: Turbulence

Table 2.3: Summary of investigations on control of secondary flows (2002 – 2014).

order. Details of each investigation are presented under five main column headings: Author(s), Type of Study, Measurement Details, Secondary Flow Control Method, and Flow Conditions. Within the Measurement Details column, the location and the type of measurements taken are identified. Within the Secondary Flow Control Method column, a check mark indicates the method investigated (i.e. one of the five methods mentioned). Abbreviations used in the tables are listed directly under the tables. Note that some investigations examined two methods within the same study (i.e. each method applied to a different cascade or test rig); those investigations will have more than one check mark corresponding to the methods studied. Several investigators have used two or more methods in combination (i.e. applied to the same cascade or test rig simultaneously), this is indicated by more than one check mark *with* a + sign beside it. With reference to the tables, a discussion of each of the flow control methods along with a summary of principal investigations relevant to it follows.

2.5.2 Endwall Fencing Investigations

Endwall fencing implements the use of endwall “fences” or guards, to control or limit the movement of secondary flows in such a way as to reduce their strength and associated losses by controlling their trajectory through the blade passage. Chung *et al.* (1991), and Chung & Simon (1993) implemented a single endwall fence with a triangular cross section, and a height of 4.5% of chord in a single passage linear cascade. They implemented the fence near mid-passage to prevent the migration of the pressure side leg of the horse-shoe vortex towards the suction side of the blade passage. They found that the passage vortex was weakened and as a result total losses were reduced by about 15%. Aunapu *et al.* (2000) uses the same location to introduce blockage in the form of rows of jets to achieve the same “fencing” effect to control the trajectory of the pressure side leg of the horse-shoe vortex. The jets were successful in deflecting the migration of the passage vortex towards

the suction surface, however, the passage vortex was not significantly weakened, and the overall secondary losses increased as a result of the losses introduced by the jets.

2.5.3 Three Dimensional Blade Investigations

Three dimensional blade designs including lean, sweep, taper, twist, and blended profiles, can be used in low-pressure (LP) turbines that usually have high aspect ratio blades to control the spanwise variation of stage reaction in actual engines. In high-pressure turbines (HP), where blades tend to have low aspect ratios, and thus the flow is dominated more by secondary flows, three dimensional blade designs can be used to reduce secondary losses, or when applied to stator blades, to improve inlet flow to downstream rotor blade rows.

Harrison (1990) provides a comprehensive investigation into the performance of leaned blades, comparing measurements on three linear cascades: a baseline straight cascade, a simple lean cascade, and a positive compound lean cascade (i.e. bowed with concave suction side). He concluded that simple lean reduced losses substantially at one endwall (lower pressure end) and increased them at the opposite wall with no net gain, while compound lean can reduce secondary flow losses but at the expense of an increase in profile losses. Thus, compound lean had little if any effect on the overall row losses but can considerably improve exit flow angle uniformity that could potentially be beneficial in reducing losses in downstream rows in real machines. Han *et al.* (2002) provide a substantial amount of flow visualization on straight, and negative compound lean (i.e. bowed with convex suction side). Bagshaw *et al.* (2005) presents a detailed experimental investigation into the effects of what they refer to as reverse compound lean (simply another name for negative compound lean). They found that their reverse compound lean cascade resulted in an increase in the secondary flow losses near the endwall, with a substantial reduction in the midspan losses. The decrease in midspan losses made up for the penalty observed in the secondary losses with an overall reduction in loss by 11%. Most recently,

D'Ippolito *et al.* (2011) examined simple lean, comparing results to a baseline non-leaned blades in both annular and linear cascades, but with no conclusive results on loss reduction.

2.5.4 Leading Edge Modification Investigations

Leading edge modifications aim to positively control the formation and evolution of the horse-shoe vortex at the leading edge where it originates by means of endwall-leading-edge fillet or bulb geometries. Sauer *et al.* (2001) presents experimental results showing the influence of several leading edge bulb geometries on the secondary flow and its losses using low speed linear cascades. They reported a remarkable 50% reduction in secondary flow losses. However, they did not present any total row losses. They attributed this reduction in loss to an increase in the intensity of the suction side leg of the horse-shoe vortex (caused by the leading edge bulb), such that it would prevent the impingement of the passage vortex on the suction side of the blade. Zess & Thole (2002) provide a similar example where a leading edge fillet was used to eliminate the horse-shoe vortex and produced significant reductions in streamwise vorticity. The study, however, was carried out using laser doppler velocimetry, with no pressure probe measurements, and thus no loss results were obtained. More recently, Becz *et al.* (2004) presented experimental results for three different cascades; a baseline cascade, a cascade with a leading edge bulb, and a cascade with a leading edge fillet. Their results showed a reduction of 7% in overall losses for the fillet, and no change for the leading edge bulb. They also found that for the cascade with the fillet the overall flow turning was slightly reduced, while for the cascade with the bulb it increased slightly.

2.5.5 Axisymmetric Endwall Contouring Investigations

Axisymmetric endwall contouring modifies the geometry of the endwall inside the blade passage only in the axial direction, with no change in the endwall geometry in the pitchwise direction. The goal is generally to locally accelerate or decelerate the flow to modify the

endwall boundary layer development with the aim of lowering secondary losses. Deich *et al.* (1960) reported on a large volume of work investigating the effects of axisymmetric endwall profiling in both linear and annular cascades. The profiles used reduced the velocity in the region of highest turning and increased the acceleration upstream of the trailing edge, with a 1.5% reduction in overall loss achieved. Since then, Kopper *et al.* (1981), Boyle & Haas (1982), Boletis (1985), and Moustapha & Williamson (1986) have all reported on using axisymmetric endwall contouring with varying degrees of success. More recently, Sonoda *et al.* (2007) presents a computational study employing axisymmetric endwall contouring reporting a total pressure loss reduction of up to 10% with contouring applied to both hub and tip endwalls.

2.5.6 Non-Axisymmetric Endwall Contouring Investigations

Non-axisymmetric endwall contouring, has recently begun to be used to control the cross-passage pressure gradient in the blade passage in a favourable manner as to reduce the secondary losses. The cross-passage pressure variation is altered by means of streamline curvature. The contouring, which varies in height both axially and pitchwise, employs convex and concave curvatures, to accelerate or decelerate the endwall flow locally with resultant decreases or increases in the local static pressure. By combining the convex and concave curvatures, the contouring in effect controls the cross-passage pressure gradient which can then be used to control the secondary flow within the blade passage to lower the secondary losses.

While non-axisymmetric endwall contouring has only relatively recently gained attention in the literature, it has been examined as early as the mid 70s. Morris & Hoare (1975) and Atkins (1987) have both reported on the use of non-axisymmetric endwall contouring; however, both failed to reduce losses using such contouring. The successful application of non-axisymmetric endwall contouring continued to elude investigators until the late 1990s.

This success can be traced back to the work of Rose (1994) who described a computational fluid dynamics (CFD) study into the use of non-axisymmetric endwall contouring to reduce static pressure variations in the flow field downstream of a stator. The objective behind this sought-after reduction, was to minimize the required coolant flow for the downstream rotor row, thus improving the overall machine efficiency. Rose's contouring used the principle of streamline curvature, employing convex and concave curvatures, to accelerate the endwall flow, or diffuse the flow causing a drop or increase in the local static pressure respectively. Rose's idea was implemented experimentally by Hartland *et al.* (1998) in a linear cascade and the contouring was successful in reducing static pressure variations at exit from the blade row.

This same approach of using streamline curvature was successfully applied to modify the cross-passage gradient in the blade passage with the aim of reducing secondary flow losses. Harvey *et al.* (2000) describes the design of the non-axisymmetric contouring, while Harland *et al.* (2000) reports on the testing. The contouring was applied upstream and downstream as well as inside of a blade passage in a linear cascade, resulting in an overall loss reduction of 20%. Gregory-Smith *et al.* (2001) reports on the testing of another linear cascade using the same method described by Harvey *et al.* (2000), but with an endwall contouring design limited to only within the blade passage. This limited contouring, supposedly easier to implement in actual engines according to the authors, resulted in a substantially lower loss reduction of 5%. Since then, this method of non-axisymmetric endwall contouring was applied to actual engines. Brennan *et al.* (2001) describes the design process for the implementation of the endwall contouring in the high-pressure (HP) turbine stage of the Rolls-Royce Trent 500 engine. Rose *et al.* (2001) reports on the testing of this design, reporting a $0.59\% \pm 0.25\%$ increase in the HP turbine stage efficiency. Finally, Harvey *et al.* (2002) reports on the implementation of the endwall contouring in a multi-row environment where it was implemented on both the HP and the intermediate-pressure

(IP) turbines of the Rolls-Royce Trent 500 engine with a reported increase of $0.9\% \pm 0.4\%$ in the stage efficiency of the IP turbine.

The development of non-axisymmetric endwall contouring requires parametric-based techniques for defining the endwall geometry, and the application of these techniques within an optimization framework to determine the best endwall shapes for a given flow configuration. The methodology presented by Harvey *et al.* (2000) employs a streamwise b-spline in conjunction with multiple pitchwise Fourier series based curves. Since then, Nagel & Baier (2005) presented a method employing the superposition of axisymmetric and non-axisymmetric shape functions, while Saha & Acharya (2006) used a combination of streamwise and pitchwise shape functions that when multiplied together and scaled, defined a new endwall geometry. Most recently, and most relevant to this study, Praisner *et al.* (2007) presented a method that used two-dimensional cubic splines in the streamwise and pitchwise direction to define the endwall contouring. As this was the method used to generate the endwall contouring used in the present study, further details are provided in the following subsection.

2.5.7 Non-Axisymmetric Endwall Contouring Used in Current Investigation

The method used to generate the non-axisymmetric endwall contouring used in the present study was developed by Pratt & Whitney Aircraft (PWA) and is described by Praisner *et al.* (2007). Their method uses two-dimensional cubic splines in the streamwise direction, along with a sinusoidal function in the pitchwise direction at the throat line of the blade passage. The splines are controlled by an array of control points distributed along the endwall as shown on the left side of Figure 2.2. Each control point is limited to one degree-of-freedom, allowing only perturbations up or down in a direction normal to the endwall. Additionally, in the pitchwise direction, a correction scheme is applied in the

form of a sinusoidal function to adjust the endwall contour to maintain the flow area in the throat region.

Figure 2.2: Control points for generating contoured endwall geometry (left side), and two sample generated geometries (right side), adapted from Praisner *et al.* (2007).

The contouring domain is defined axially between the leading and trailing edges of the airfoil (i.e. does not extend upstream or downstream), with a population of five equally spaced axial control points, with five pitchwise points per axial row. The first and last column of control points at the leading edges and the trailing edges respectively, are fixed at the nominal endwall height, leaving the remaining points to be perturbed individually. Several permutations (in the order of 1000) of possible perturbations are generated, and solved using an in-house 3D structured Reynolds Averaged Navier Stokes (RANS) code, using a design level (i.e. coarse) mesh for quick convergence. The optimization is done using a Sequential Quadratic Programming (SQP) gradient-based algorithm, where the converged solution of the flat endwall is used as the initial solution for the domain. The optimization criterion that is used is reduction of mass-averaged total row loss, with the additional constraint that the local exit flow angle deviation near the endwall (deviation

from the midspan flow angle at about 5% span) not increase. The right side of Figure 2.2 shows sample converged solutions of the resulting endwall geometry. Praisner *et al.* (2007) also reported experimental linear cascade results employing this method of endwall contouring showing reductions in overall loss of 10% and up to 25% in some cases. Knezevici *et al.* (2010) describes the experiment in more detail, shedding light on some of the flow physics involved.

The implementation of non-axisymmetric endwall contouring has been successfully demonstrated by several investigators. However, CFD is central to all methods used in generating such complex and optimized geometry, and thus experimental verification of predicted performance is needed. While CFD may be trend accurate enough for optimizations studies, there are cases where it can fail to capture real physical flow phenomena that can lead to a significant change in trend (e.g. increase in losses and failure of endwall contouring). Ingram *et al.* (2005) reports on such a case, where an implementation of an aggressive non-axisymmetric endwall contouring resulted in an increase in secondary losses contrary to the CFD predicted benefits. The authors attributed the discrepancy to a local flow separation on the early suction side of the passage that was not captured by CFD. The authors conclude that improvements in CFD capability are required in order to avoid such flow features and obtain the full potential of endwall contouring. Such conclusions highlight the importance of, and need for, experimental validations, such as the ones in this present work, of new concepts and design methodologies.

2.5.8 Mixed Methods Investigations

Several secondary flow control methods have been discussed. However, investigators since the late 1990s have been examining the feasibility of combining two or more of these methods. Duden *et al.* (1999) provides an early example of this (three-dimensional blade design and axisymmetric endwall contouring). A more recent example is given by Nagel

& Baier (2005), where a full optimization was carried out on the blade passage resulting in highly irregular looking three dimensional blade designs coupled with relatively familiar looking non-axisymmetric contoured endwalls resulting in secondary flow reductions in both numerical and experimental studies. Most recently, Gregory-Smith *et al.* (2008) reports on using a combination of non-axisymmetric endwalls, blade lean, and leading edge modifications to reduce secondary flow losses.

In this present study, in some respects, a mixed method of flow control is employed. Early in this section, it was mentioned that one way to reduce the secondary flow losses is by having a good two dimensional profile design before three dimensional blade or endwall modifications are considered (e.g. blade lean or endwall contouring). For example, several researchers have confirmed that aft-loaded profiles can produce significantly lower secondary losses than the front-loaded designs. A recent example demonstrating this is given by Popovic (2005). In this present study, in addition to the non-axisymmetric endwall contouring, the blade cascade geometry used (designated SL2P) was developed using CFD by PWA specifically to reduce the secondary flow losses by modifying the pressure side geometry of a given airfoil to produce a favourable blade loading distribution. Taremi (2011) provides detailed experimental results using this airfoil on a flat endwall with comparisons to its non-modified counterpart (designated SL2) as part of his doctoral thesis work.

2.6 Investigations on Upstream Cavity Effects

In actual gas turbine engines, there is always a change in the endwall geometry upstream of the rotor in the form of a cavity (see Figure 1.2). This cavity is necessary for mechanical design considerations as well as cooling considerations as discussed earlier. The horse-shoe vortex discussed in Section 2.2 originates from the endwall boundary layer at the leading

edge, and thus an upstream cavity can have a significant impact on the evolution of the secondary flows and the secondary flow losses.

There is a limited amount of investigations related to upstream cavity effects in the open literature. The majority of investigations focus on sealing effectiveness, and purge flow effects, with little attention to how changes in the upstream cavity geometry can effect the flow physics. Nonetheless, a summary of relevant investigations—some more pertinent than others—is provided in Table 2.4 in chronological order. Within the Measurement Details column, the location and the type of measurements taken are identified, as well as whether purge flow or secondary flow control methods (e.g. endwall contouring) were used. The amount of purge flow used in the study in percentage of total mass flow is given under the flow conditions.

Early investigations focused on design of sealing and cooling air systems. An early and thorough example is provided by Campbell (1978). Later papers, like the work of Kobayashi *et al.* (1984), and Daniels *et al.* (1992) focus on evaluating the sealing effectiveness of different seal configurations. Since then several studies have examined the effects of purge flow on the mainstream flow. Hunter & Manwaring (2000), Anker & Mayer (2002), McLean *et al.* (2001a), and Paniagua *et al.* (2004) have all demonstrated that purge flows strengthen the endwall secondary flows downstream of the rotor blades in rotating test rigs.

A few papers have focused exclusively on the upstream cavity geometry itself. Wu & Zhong (2003) examined qualitatively the effects of a forward facing step on the secondary flow in a vane passage. They used multiple smoke wires to visualize the structures near the endwall of an enlarged turbine vane at low speeds. They found that the forward facing step introduced a multiple vortex system directly after the step instead of the usual horse-shoe vortex. de la Rosa Blanco *et al.* (2005) examined the effect that upstream endwall geometry can have on the secondary flows in a low-speed low-pressure turbine linear cascade,

Author(s)	Type of Study	Measurement Details						Flow Conditions						
		Within Passage	D/S of TE	Blade Surface	Pressure Probe	Hot Wire / Laser	Flow Control	Purge Flow	Flow Vis.	Γ (°)	$Re \times 10^5$	M_2	Tu (%)	Purge Flow %
Campbell (1978)	Engine							✓		-	-	-	-	-
Kobayahsi et al (1984)	RTR	✓						✓		design	60 (rotational)	-	-	0.065-0.204 (kg/s)
Daniels et al (1992)	RTR	✓						✓		design	51 (rotational)	-	-	-
Burd & Simon (2000)	LC	✓	✓			✓	✓	✓		design	3.5	-	9	0 - 6
Hunter & Manwaring (2000)	RTR		✓		✓	✓		✓		design	4.5	design	-	-
Kost & Nicklas (2001)	LC	✓	✓		✓	✓		✓	oil	design	8.5	1	-	1.3 (slot), 0.32 (holes)
McLean et al (2001a)	RTR		✓		✓			✓		design	3.5	-	-	1, 1.25, 1.50
McLean et al (2001b)	RTR		✓		✓			✓		design	3.5	-	-	1
Anker & Mayer (2002)	CFD/RTR		✓		✓			✓		design	4	0.15	-	4.6
Cao et al (2003)	CFD/RTR	✓	✓		✓			✓		design	60 (rotational)	-	-	1.09
Gentilhomme et al (2003)	CFD/RTR		✓					✓		design	6.23, 1.77	0.963, 0.390	-	-
Wu and Zhong (2003)	LC					✓			smoke	design	0.34	-	-	-
Beaudoin et al (2004)	Step	✓				✓			oil	-	-	-	-	-
Paniagua et al (2004)	CFD/RTR	✓	✓	✓	✓			✓		design	5	0.89	-	-0.5, 0.6, 1.5
Cherry et al (2005)	CFD									design	-	-	-	-
de La Rosa Blanco et al (2005)	LC/CFD		✓		✓				oil	design	2.32	0.07	0.5	-
Reid et al (2006)	RTR/CFD		✓		✓			✓		design	-	-	-	0-3
Marni & Grgis (2007)	CFD							✓		design	1.2	1.15	-	0, 0.4, 0.8, 1.2
Popovic & Hodson (2010)	LC/CFD		✓	✓	✓			✓	oil	design	5	-	0.5	0.5, 1, 1.5
Schuepbach et al (2010)	RTR/CFD		✓		✓		✓	✓		design	-	-	-	0.5, 0.9, 1.3
Schuler et al (2010)	CFD							✓		design	5.9	-	-	0, 0.5, 1
Schuepbach et al (2011)	RTR/CFD		✓		✓		✓	✓		design	5.1	0.46	-	-0.1, 0.9
Schuler et al (2011)	LC		✓	✓	✓	✓		✓		design	5.6	0.22	5.88	0, 1
Zlatkov (2011)	CFD							✓		design	-	-	10	0.5

Abbreviations:

LC: Linear Cascade
RTR: Rotating Test Rig
CFD: Computational Fluid Dynamics
D/S: Downstream
TE: Trailing Edge

Vis.: Visualization

Γ : Incidence

Re: Reynolds Number

M_2 : Outlet Mach Number

Tu: Turbulence

Table 2.4: Summary of investigations on upstream cavity effects.

both experimentally and numerically. They investigated two simple geometries upstream of the blade row, forward facing and backwards facing steps, with several variations in the step height and the thickness of the inlet endwall boundary layer. They reported a three-dimensional separation bubble behind the backwards facing step and that the fluid entrained into the bubble ends up being part of the passage vortex. Overall, they found that the presence of the step upstream can significantly alter the structure and the strength of the secondary flow, with certain step heights and step orientation providing more favourable results in terms of losses than others.

Some papers addressed the effects of different geometries while also evaluating the effects of purge flow. Marini & Girgis (2007), presented a computational study of a transonic turbine stage that included the upstream cavity. In their study, they investigated two different rotor leading edge platform geometries, one with a recessed platform, and another with a raised platform, resulting in two distinct cavity geometries. They examined these two geometries with no purge flow, as well as at several engine representative purge flow rates. They found that the cavity geometry formed by the raised leading edge platform shape showed an improvement in stage efficiency as well as reduced sensitivity of stage efficiency to purge flow rates. More recently, Schuler *et al.* (2011) examined two different rim seal platform overlap geometries in a low-speed linear cascade with and without purge flow. They concluded that the mere presence of the seal geometry leads to an increase in total pressure losses, with the addition of purge flow further increasing the losses.

With the reported success of endwall contouring, some researchers have looked to examine its effectiveness with upstream cavity geometry and purge flow. A recent example is provided by Schuepbach *et al.* (2011), where they presented rotating test rig and numerical results using non-axisymmetric contouring applied to the endwall. The authors concluded that the endwall contouring performance deteriorated with the addition of purge flow.

2.7 Previous High Speed Investigations at Carleton University

The High Speed Wind Tunnel at Carleton University has been used extensively in the past for midspan aerodynamic measurements in transonic axial turbine cascades. The primary focus of the early research was on profile loss measurements using three-hole pressure probes. The high speed wind tunnel, was first commissioned in 1994, and was then used extensively for profile loss measurements by Islam (1999), Jeffries (2000), Jouini (2000), and Corriveau (2005). Various high pressure and low pressure turbine cascades have been examined in order to study the effects of compressibility and off-design incidence on profile losses.

Recently, the test section of the wind tunnel was modified to allow for three-dimensional flow measurements using a seven-hole pressure probe. A summary of this modification can be found in Taremi *et al.* (2008). This modification was made to accommodate the testing of several Pratt & Whitney USA cascades for the investigation of secondary losses. The most recent results of this investigation are given by Taremi *et al.* (2011). Complete details of this test section and the wind tunnel are presented in the next chapter.

2.8 Summary and Conclusions

The literature review presented in this chapter showed that the majority of the research on secondary flow focused on low-speed flows with only a handful examining high-speed flows representative of HP turbines. Thus, there is strong motivation to expand the understanding of the flow physics to the compressible regime (HP representative speeds).

Limited research exists on the effects of upstream cavity on the secondary flow. Further, these limited investigations suggested that simple steps or cavities upstream of a cascade

can have a very large impact on the secondary flow and losses. Therefore, investigating the effects on the secondary flow of a cavity that approximates the gap between the rotor and stator (as was done in this study) would be a significant contribution.

Finally, several non-axisymmetric endwall contouring designs were shown to be viable options for reducing secondary losses by previous researchers. However, the performance of the PWA non-axisymmetric endwall contouring design has only recently been investigated experimentally at high-speeds, and further studies are needed to assess its viability and robustness. Thus, in this present study, the PWA contouring design is assessed and studied with a modern high-lift airfoil (SL2P) and an upstream cavity, at both design and off-design Mach numbers.

Experimental Method and Data Reduction

3.1 Introduction

In this chapter, the high speed wind tunnel used in this study is first presented. Description of the test section and details of the measurements and the instrumentation employed follows. A new addition to Carleton's high speed facilities, the High Speed Probe Calibration Rig, also used in this study, is then described. Next, the test cascade is presented followed by a description of the flow visualization technique. Finally, the data reduction methods for all measured quantities are presented in the last section.

3.2 High Speed Wind Tunnel

3.2.1 Overview of the Wind Tunnel

The Pratt & Whitney Canada (PWC) High Speed Wind Tunnel, located in the Carleton University High Speed Aerodynamic Laboratory (HSAL), was first commissioned in 1994, and has since been used extensively for transonic turbine cascade testing.

Figure 3.1: Pratt & Whitney Canada High Speed Wind Tunnel.

The wind tunnel, illustrated in Figure 3.1, is of the blow-down type with run durations in the range of 30 to 60 seconds depending on the outlet Mach number of the test cascade.

The outlet Mach number can be varied from about 0.3 to 1.5. Four storage air tanks with a total volume of about 26 m³ supply the wind tunnel air. The storage tanks are fed by a large Broom-Wade VC500 reciprocating air compressor, pressurizing the tanks to approximately 700 kPa(g). After exiting the compressor, the air is passed through an after-cooler bringing the air temperature down from about 100 °C to about 27 °C. Prior to storage in the tanks, moisture in the air is removed by a water separator, and a Comp-Air Kellogg CDH520 regenerative desiccant dryer reduces the dew point of the air to below −45 °C to prevent the formation of condensation in the wind tunnel.

Once the tanks are fully pressurized, the wind tunnel shutoff valve is opened and the air is discharged into the test section through the wind tunnel control valve; both valves shown in Figure 3.1. The wind tunnel control valve controls the blowing pressure based on static pressure measurements made by an Omega PX613 pressure sensor with a range of 100 psi(g) located 54.0 axial chords (C_x) upstream of the cascade leading edge. The freestream turbulence intensity at the inlet of the cascade test section is approximately 4% with a turbulence length scale of about 15 mm (Corriveau, 2005). The calibration rig shutoff and control valves also shown in the figure, are used for the probe calibration rig, which is discussed in Section 3.4. Figure 3.1 shows a 183 cm (6 ft) person for scale, and highlights the wind tunnel test section. The test section is presented in detail next.

3.2.2 Wind Tunnel Test Section

The wind tunnel test section, illustrated in Figure 3.2, can accommodate a linear cascade of six to seven blades (depending on the cascade solidity) with chord lengths ranging from about 30 to 45 mm, and a span of 61 mm. Typically, six or seven blades with two matching end blocks are mounted on an endwall panel forming the cascade. The endwall panel is then mounted inside the test section. The panel fits into a matching pocket flush with the test section's bottom wall.

A special cobra probe using a pitot tube combined with a thermocouple is mounted 49.0 C_x upstream of the cascade leading edge to provide reference total pressure and total temperature values. A window on top of the cascade provides optical access to the test section. A window on top of the cascade provides optical access to the test section.

Figure 3.2: Wind tunnel test section.

The test section allows for measurements upstream and downstream of the cascade. Downstream of the cascade, as illustrated in Figure 3.2, a stepper motor powers the traversing gear to move the probe stem in the pitchwise direction. The stepper motor is capable of 200 steps per revolution with a resultant pitchwise displacement of the traverse gear

of 1.95 mm per revolution. The probe stem enters the test section through a sealed slot passing through the outlet walls. This slot is sealed using a tight fitting teflon block. In the spanwise direction, the traversing gear is adjusted manually, lifting or lowering one end of the probe stem while the other end is supported by spacers that fit above or below the teflon seal piece depending on the required spanwise location. At the end of the probe stem is the probe holder, which can accommodate up to two probes simultaneously. In this study, however, only one probe was mainly used; a seven-hole pressure probe. Upstream of the cascade, a resealable hole allows for a pitot probe of the goose-neck design to be inserted to traverse the boundary layer using the same stepper motor for the downstream measurements, but mounted vertically in a different traversing gear to allow traversing in the spanwise direction. The test section also allows for additional static pressure, total pressure, and total pressure measurements at several locations upstream and downstream of the cascade using several instruments. Further details of the measurement locations, and instrumentation are presented in the next section.

3.3 Wind Tunnel Measurements and Instrumentation

3.3.1 Measurement Locations and Procedures

The testing procedure for the cascade involved four main steps: establishing the blowing pressure, flow quality checks, pitot pressure probe boundary layer traverse, and seven-hole pressure probe full plane traverses. Figure 3.3 shows the measurement locations. From upstream to downstream these include: a total pressure and total temperature measurement (cobra probe with Type K thermocouple located at $49.0 C_x$ upstream of the leading edge), pitot probe boundary layer measurements, inlet static pressure measurements, blade loading measurements, outlet static pressure measurements, and lastly, seven-hole pressure probe measurements.

Figure 3.3: Experiment measurement locations.

The testing procedure for the test cascade starts with establishing the blowing pressure for the operating point corresponding to the design Mach number of 0.80 of the blade profile. The blade profile was designed using CFD, thus the wind tunnel blowing pressure for the design condition was adjusted such that the experimental blade loading matched the computational blade loading values at design. Experimental blade loading values are provided by blades 3 and 5, which are instrumented with static taps on the pressure side and suction side respectively, as shown in Figure 3.3. Good agreement between computational and experimental loading was found at a blowing pressure corresponding to the design outlet Mach number of 0.80. The blowing pressure can then be varied to obtain off-design Mach numbers.

Once the blowing pressure is established, and the blade loading measurements taken, the two instrumented blades are replaced with non-instrumented blades. Flow quality checks are then performed. The row of upstream static taps was used to confirm inlet flow uniformity shown in Figure 3.4. The downstream static taps were used to confirm the outlet flow periodicity, and a further periodicity check was done by traversing the wakes of blades two to six at 50% span using a seven-hole pressure probe as shown in Figure 3.5. With the flow quality established, the upstream boundary layer was then traversed at $2.0 C_x$ upstream of the leading edge of the cascade using a goose-neck pitot probe with a diameter of 0.9 mm.

Figure 3.4: Typical inlet uniformity, cavity endwall measurements shown.

Finally, seven-hole probe measurements were taken at $1.4 C_x$ from the leading edge. The measurements were taken for half the blade span at 16 spanwise locations spanning from approximately 2.5% to 50% of the span. At each spanwise location, the full pitch of the 4th blade wake was traversed at 2.5% intervals of the blade pitch. The measurement grid shown in Figure 3.3 is denser closer to the endwall to capture more of the endwall flow structures. The design of the seven-hole probe is described in Section 3.3.2, while the

Figure 3.5: Typical wake periodicity at 50% span, cavity endwall measurements shown.

pressure transducers used to capture these measurements are discussed in Section 3.3.3. The use and reduction of these measurements is discussed in Section 3.7.

The above procedure was applied to all three endwalls; flat endwall, cavity endwall, and contoured endwall. All three test endwalls were tested at design incidence and design Mach number of 0.80. At this condition, the Reynolds number based on outlet velocity and axial chord is about 600,000. Additionally, all three endwalls were tested at off-design Mach numbers of about 0.69, 0.75, 0.84, and 0.89. Since Mach number was varied by changing the blowing pressure, this also resulted in variations in the Reynolds number from about 500 000 to 700 000. Surface flow visualization was also carried out on each build at the design Mach number of 0.80; details of the technique are presented in Section 3.6.

3.3.2 Seven-Hole Pressure Probe

A seven-hole pressure probe with an outer diameter of 1.83 mm was used for detailed downstream measurements of the cascade. This seven-hole probe was originally used by Knezevici (2011) to study secondary flows.

The probe design shown in Figure 3.6, is of the goose-neck type to allow measurements near the endwall. Prior to use in the wind tunnel, the probe was calibrated at Mach numbers, 0.48, 0.67, 0.77, and 0.95. Calibration was done over the angle range of $\pm 20^\circ$ in 2° increments in both pitch and yaw using the high speed probe calibration rig (later discussed in Section 3.4). For each of the pitch and yaw angle positions (α and β), the seven port pressures of the pressure probe were used to calculate the calibration coefficients:

$$C_\alpha = \frac{2(P_4 - P_1) + P_3 - P_6 - P_2 + P_5}{3(P_7 - \bar{P})} \quad (3.1)$$

$$C_\beta = \frac{P_3 - P_6 + P_2 - P_5}{\sqrt{3}(P_7 - \bar{P})} \quad (3.2)$$

$$C_o = \frac{P_7 - P_o}{P_7 - \bar{P}} \quad (3.3)$$

$$C_q = \frac{P_7 - \bar{P}}{P_o - P_s} \quad (3.4)$$

where

$$\bar{P} = \frac{P_1 + P_2 + P_3 + P_4 + P_5 + P_6}{6} \quad (3.5)$$

These calibration coefficients were calculated for each of the four calibration Mach numbers (0.48, 0.67, 0.77, and 0.95), constituting the calibration tables for the seven-hole pressure probe. These calibration tables yielded a set of calibration surfaces (functions of α and β) for each calibration coefficient at each of the calibration Mach numbers. Some of these surfaces are shown in Appendix A.

To characterize an unknown flow field, the generated calibration tables were used in an iterative solving procedure combined with an interpolating table look-up method to

extract the total pressure, dynamic pressure, Mach number and the two flow angles from the measured port pressures. Isentropic flow, and ideal gas are then assumed to obtain the values for static temperature, density, and the three components of velocity.

Figure 3.6: Seven-hole pressure probe design and dimensions.

3.3.3 Pressure Transducers

The wind tunnel pressure measurements were obtained using 11 Druck PDC-22 pressure transducers, and one Omega PX613 pressure transducer. These transducers are also used to obtain measurements on the high speed probe calibration test rig (later discussed in Section 3.4). Each of the transducers is connected to a dedicated channel on the data acquisition and control systems (described in the following section). Part number, range, and function of each transducer are tabulated in Table 3.1. The transducers are typically calibrated twice a year using a Druck DPI 605 Pressure Calibrator. Additionally, daily

checking of the transducers wind-off values was done prior to tunnel operation. The calibration results in a linear correlation relating the voltage output to measured pressure for each transducer. These correlations are provided in Appendix B. The Druck DPI605 Pressure Calibrator itself was calibrated in May 2010 using a US Airforce Standards calibration method (33K6-4-427-1), traceable to an NIST (National Institute of Standards and Technology) standard. During normal course of operation, the calibrator does not require recalibration (Druck, 1994).

Table 3.1: Pressure transducers.

Channel No.	Serial No.	Range (psi)	Type	Measured Pressure
1	134061	100	Differential	Probe port 1
2	B6228	50	Differential	Probe port 6
3	82842/834	50	Differential	Reference total
4	114370	50	Differential	Probe port 7
5	B6992	100	Differential	Probe port 5
6	B3116	75	Differential	Inlet static
7	76741	100	Absolute	Probe port 4
8	76740	100	Absolute	Probe port 3
9	134060	100	Differential	Outlet static
10	B6202	100	Differential	Probe port 2
11	76738	50	Absolute	Atmosphere
A	Omega PX613	100	Differential	Blowing pressure control

As noted in Table 3.1, nine of the 12 transducers are of the differential type. To obtain absolute values from these differential transducers, one end is left open to atmosphere, atmospheric pressure is then measured by an absolute transducer (Channel 11) and then added to the differential transducers measurements. All of the Druck transducers are placed in an insulated container to ensure that they are all exposed to the same atmospheric pressure, and to minimize any variations in ambient temperature.

3.3.4 Control and Data Acquisition Systems

The control and data acquisition system for the wind tunnel achieve several functions: control of the blowing pressure valve, control of the traversing gear stepper motor, acquiring of pressure measurements, and acquiring of temperature measurements.

The first function, i.e. control of the blowing pressure, is accomplished by an independent system from the rest of the other functions. This system is thoroughly documented in Corriveau (2005). In brief, it is comprised of an Intel Pentium 90 PC equipped with a Hewlett-Packard 3852A data acquisition and control unit. This system acquires pressure measurements from an upstream Omega transducer (see Table 3.1) to be used in a proportional integral derivative (PID) feedback loop. This PID feedback loop outputs positions to the wind tunnel control valve to achieve a user specified pressure value. For a typical Mach number of 0.80, the control system can ramp up and reach the target Mach number in about 20 seconds. The controller can then maintain the Mach number within ± 0.005 of the target value. Typical run durations after ramp up, range from about 30 to 60 seconds depending on the target Mach number.

The other three functions, i.e. control of the pitchwise traversing gear stepper motor, acquiring of pressure measurements, and acquiring of temperature measurements, are accomplished through a second independent system. Stepper motor control is achieved using an in-house designed motor controller connected to an Intel based Microsoft Windows PC via the parallel port. A National Instruments PCI-6229 sixteen-bit data acquisition card connected to the same Microsoft Windows PC via the PCI interface is used to acquire measurements from the 11 Druck pressure transducers to obtain the seven-hole probe pressures as well as reference static and total pressures. These eleven Druck pressure transducers were described in the previous section. The same PC also accommodates a National Instruments USB-9211A Thermocouple Input Module connected via a USB port to acquire total temperature values from the Type K thermocouple. A software developed

under LabVIEW 8 manages the collection of the pressure and temperature data as well as control of the stepper motor. The software records the average of 200 samples for each measurement, obtained at a sampling rate of 2000 Hz. These values were chosen based on an investigation carried out by Jeffries (2000) on the effect of sampling time and frequency on the accuracy of measurements in the wind tunnel.

In a typical run, once the operating Mach number is established using the first system (blowing pressure control system), the second system (controlling the stepper motor, and acquiring the pressure and temperature measurements) is initiated by the user. Once the traversing is complete, the user then shuts down the wind tunnel using the first system. This procedure can then be repeated for the next spanwise traversing location.

3.4 High Speed Probe Calibration Rig

3.4.1 Preliminary Design

The non-nulling mode, where the probe orientation is held fixed relative to an unknown flow field, is typically used in three-dimensional flow measurements, as is the case in this study. In order for a pressure probe to be used in the non-nulling mode, it must first be calibrated in a flow of known direction. This can be achieved by aligning the probe in a jet which provides a uniform flow, with a probe holder that allows for rotation of the probe while keeping it's mouth fixed at the same location.

Prior to 2006, probe calibrations were carried out in the high speed wind tunnel test section. This required repeated cumbersome disassembly of the test section, was both time consuming and labour intensive, and limited the range of flow angles relative to the probe that could be achieved. This prompted the development of a dedicated probe calibration rig with the capability of calibrating probes at high speeds. To achieve this, the design required that a uniform jet be generated, and that the probe mounting

system allows for probe axial rotations in 90° steps for probe aerodynamic alignment, while keeping the mouth of the probe fixed at the same location. The design was to allow for rotation of the probe in both the pitch and yaw directions relative to the jet over a range of $\pm 35^\circ$. Further design constraints included that the calibration test rig make use of existing air supply and control valves and allow for a calibration range of about Mach 0.2 to 0.95. Preliminary design work was started in the summer of 2006 by Andrew Shearer (undergraduate student). Since then, the present author has completed the final design, including shake-down of the test rig, development of the motor control and data acquisition system, and a full calibration of the seven-hole pressure probe used in this study. A brief summary of the latter work is provided in the following sections.

3.4.2 Final Design

The final design of the High Speed Probe Calibration Rig is illustrated in Figure 3.7. The rig makes use of the same air supply system used for the wind tunnel but uses separate shutoff and control valves to the ones used for the main wind tunnel (see Figure 3.1). The control valve provides air to the test rig through hoses connected to both sides of the mixing chamber. As the air enters from both sides, a bell-mouth guides the air to a straight nozzle to provide the calibration jet, which is discharged to the atmosphere. One of two static taps on the mixing chamber (where the air settles before accelerating through the bell-mouth inlet) provides a reference total pressure value. The second static tap on the mixing chamber is used to control the blowing pressure of the jet using the same Omega transducer and valve control software used for the wind tunnel. Two static taps at opposite ends of the exit of the nozzle give equal values as expected for a uniform jet. One of the two static taps on the nozzle is used as a reference static pressure value. These reference static and total pressure values were used in conjunction with a pitot-static pressure probe to provide correction factors for establishing the jet's true operating point.

Figure 3.7: High speed probe calibration rig.

The probe platform, directly downstream of the jet, is capable of positioning a probe over a range of $\pm 35^\circ$ in the pitch direction, and $\pm 30^\circ$ in the yaw direction. This is accomplished by two Vexta PK245-01AA stepper motors; one for pitch, and one for yaw. These motors are combined with Velmex A5990TS rotary tables giving them a capability of 0.01° movement per stepper motor step. Further, the probe holder allows for probe rotations in 90° steps while keeping the face of the probe fixed at the same location. This allowed for proper aerodynamic alignment of three-hole and seven-hole pressure probes relative to the calibration jets. The probe holder can also be shifted laterally, longitudinally, and axially relative to the calibration jet. This function was used in conjunction with a pitot-static probe to verify the jet flow uniformity. Steady calibration jets have been demonstrated for Mach numbers ranging from about 0.2 to 0.95.

As mentioned, measurements using a pitot-static pressure probe were done to establish the jet uniformity as well as the true static and total pressures relative to the reference values. The true static and total pressures were established at a position located one nozzle internal diameter ($1.0 D$, $D = 38 \text{ mm}$) from the centre of the nozzle at the exit plane along the x-direction (see Figure 3.7). It was found that the true static pressure to the reference static pressure ratio was 0.9850, while the true total pressure to reference total pressure ratio was 0.9804. These ratios are used as correction factors multiplied by the reference static and total pressures to obtain true values. At this same position of $1.0 D$ along the x-direction, it was found that a uniform flow region existed for all Mach numbers within $\pm 0.13 D$ from the centre of the nozzle in both the y- and z-directions. Thus, probes are calibrated at $1.0 D$, and aligned with the centre of the nozzle exit ensuring that it lies in a uniform flow with known true static and total pressures.

The seven-hole pressure probe used in this study was calibrated in the high speed probe calibration rig. Details of the design and calibration of the seven-hole probe were presented in Section 3.3.2. This particular seven-hole calibration was used in this study as well as a related study (Taremi, 2011). Subsequently, the calibration rig was used for the calibration of a three-hole probe, for measuring profile losses (Hall, 2012).

3.4.3 Control and Data Acquisition Systems

The probe calibration rig shares some of the hardware and software of the wind tunnel control and data acquisition systems. The two main differences are in the motor control hardware, and the motor control and data acquisition software.

The wind tunnel Omega transducer and valve control system usually used to control the blowing pressure in the wind tunnel, are used to control the blowing pressure for the probe calibration rig. The omega transducer is connected to the settling chamber (instead of the wind tunnel), and the valve control system is connected to the probe calibration

control valve (instead of the wind tunnel control valve). This valve control system was previously described in Section 3.3.4.

Stepper motor control is achieved using a Velmex VXM-2 two-channel motor controller connected to an Intel based Microsoft Windows PC via the RS-232 port (serial port). A National Instruments PCI-6229 16-bit data acquisition card connected to the same Microsoft Windows PC via the PCI interface is used to acquire measurements from up to 11 pressure transducers to measure probe pressures as well as reference static and total pressures. These transducers are also used in the wind tunnel measurements, and further details were provided in Section 3.3.3.

Figure 3.8: High speed probe calibration rig – motor control and data acquisition software user interface.

Motor control and data acquisition software for the high speed probe calibration rig was developed by the present author using National Instruments LabVIEW 8. This software

provides a fully automated interface to communicate with the motor controller, and data acquisition card. Figure 3.8 presents the graphical user interface of the software developed for controlling the step motors and collecting the pressure measurements. The software allows for the user to select which pressure transducers to collect measurements from, and specify the range and increment in the yaw and pitch directions. It also has a manual mode to allow the probe to be positioned to a user specified yaw and pitch angle. Under the advanced options, the user can specify the aerodynamic alignment values, sampling delay, number of samples, and sampling rate. Typically, the number of samples is 5000 samples and the sampling rate is 1000 Hz. A delay of 250 ms is usually used to allow for the probe pressures to settle at the new values before the sampling starts after each change in probe angle. These values were arrived at after an investigation carried out on the rig.

The software has a real time status display indicating to the operator the current yaw and pitch position of the probe, whether it has been initialized, whether it is currently running, and whether it is in automatic or manual mode. When in automatic mode, upon completion of the calibration, the software outputs a file containing the yaw and pitch angle, and pressure measurements, to a user specified location.

3.5 Test Cascade and Builds

3.5.1 Blade Geometry

Three builds formed by three endwalls and one common blade geometry were used in this investigation. This blade geometry, designated as SL2P, was developed using CFD by Pratt & Whitney Aircraft to reduce the endwall losses by means of modifying the pressure side surface of the airfoil as part of a related study (Taremi, 2011) on the control of secondary flows. In this present study, the SL2P geometry along with upstream cavity geometry and non-axisymmetric endwall contouring is investigated.

SL2P, is a highly loaded blade design with a Zweifel coefficient value of 1.15. The blade geometry along with relevant parameters are summarized in Figure 3.9.

Figure 3.9: SL2P – blade geometry and parameters.

This blade geometry was combined with three different endwalls to form three builds; the baseline case referred to simply as the *flat endwall*, a flat endwall with an upstream cavity (approximating the gap between the rotor and stator in actual engines as shown in Figure 1.2) referred to as the *cavity endwall*, and a contoured endwall with the same upstream cavity referred to as the *contoured endwall*. Each of the three endwalls is presented in the following sections.

3.5.2 Flat Endwall

A flat endwall, combined with the SL2P blade geometry forms the baseline case, referred to as the *flat endwall*. The endwall panel fits into the high speed wind tunnel's test section. The panel features six pockets to accommodate six blades, and it can accommodate two

end blocks, to give a total of seven passages. Additionally, it has 32 static taps located about $1.4 C_x$ downstream of the leading edge. A thorough description of the measurement locations was presented in Section 3.3.

3.5.3 Flat Endwall with Upstream Cavity

The second endwall panel adds the additional feature of an upstream cavity approximating the gap between the rotor and stator in actual engines, as shown in Figure 3.10. This endwall combined with the SL2P blade geometry is referred to as the *cavity endwall*. The cavity on the endwall panel is an approximation to an actual engine cavity (see Figure 1.2); based on the engine cavity of the HP turbine stage of an IAE V2500 turbofan engine.

The cavity starts at $0.30 C_x$ from the leading edge, extends $0.17 C_x$ in depth, and is followed by a smooth ramp to return the endwall to its nominal height as illustrated in Figure 3.10. The upstream cavity spans across all seven passages of the blade row.

Figure 3.10: Flat endwall with upstream cavity, *cavity endwall*.

3.5.4 Contoured Endwall with Upstream Cavity

The third endwall features the same upstream cavity described in the previous section, but adds non-axisymmetric endwall contouring as illustrated in Figure 3.11.

Figure 3.11: Contoured endwall with upstream cavity, *contoured endwall*.

This endwall contouring was designed computationally to minimize endwall losses including the influence of the upstream cavity. As described in Section 2.5.7, the contouring makes use of streamline curvature to control the cross-passage pressure gradient in the blade passage in a favourable manner as to reduce the endwall losses. This endwall panel combined with the SL2P blade is referred to as the *contoured endwall*. Figure 3.11 presents an isometric view of two blade passages showing the non-axisymmetric contouring and the upstream cavity (note that the upstream cavity is shown in front of only one passage for clarity), along with a contour plot showing the deviation of the endwall surface from the nominal endwall height.

3.6 Flow Visualization Technique

3.6.1 Method and Equipment

Oil surface flow visualizations were conducted on the blade suction surface, pressure surface, and endwall at design Mach number of 0.80 for all three endwalls. A fluorescent red powder pigment known as Day-Glo AX-13-5 Rocket Red, was mixed with 10W-30 motor oil in a ratio of about 1:3 by volume. This pigment comes in a fine powder form, and is classified as a synthetic organic colourant (Material Safety Data Sheet, 2007). The pigment is typically used in special effects paints or makeup kits, and can be obtained from special effects supply stores. This pigment was dissolved in the motor oil and it was found that the ratio of about 1:3 (powder to oil by volume) produced a slightly over saturated solution, giving a red oil with some undissolved pigment powder suspended in the solution. This mixture gave good results for the flow conditions used in this investigation.

The flow visualization was carried out at the design Mach number of 0.80. The test section was opened, the static pressure tapped blades were exchanged with non-tapped blades, downstream static taps located on the bottom endwall were sealed, and the bottom endwall and blades—including both suction and pressure surfaces—were painted with the oil-powder mixture using a synthetic, flat medium sized, artist oil paint brush. The application of the mixture was done fairly liberally. The test section was closed, and the wind tunnel was operated at the design condition of the cascade. Real time videos were first taken of the flow visualization during startup to determine if the transient behaviour at start up and shutdown would distort the real flow patterns. It was found that the transient effect was minimal; results from the video capture demonstrating this are given in Appendix C. It was also found, that typically 2 runs were sufficient to give a stable non-dripping flow visualization pattern. Lastly, the cascade was removed from the test section, and the resulting flow pattern was illuminated for photography in a darkened

room with ultraviolet (UV) lighting. It is worth mentioning, that the process of opening up the wind tunnel test section is a laborious and time consuming one, typically requiring a full working day; thus there were limits to the number of trials that can be done.

In order to produce effective images, the UV lighting source was to have minimal emissions in the visible spectrum, i.e. very little purple-blue light emitted. Additionally, the cascade has to be evenly illuminated to avoid bright or dark spots, and shadows. A combination of home-use fluorescent “black light” tubes and compact fluorescent “black light” lamps in the configuration shown in Figure 3.12 were used. For this setup, two Phillips 40 watt 48" black light fluorescent tubes were installed in a standard 48" overhead ballast, and two 13 watts twister medium base type compact fluorescent lamps were used. The overhead lights illuminated the object being photographed with plenty of UV light, while the two compact fluorescent bulbs were used to provide additional lighting for the target area being photographed as needed. The two compact fluorescents were kept mobile by being installed in hand held work light fixtures as shown in Figure 3.12. This allowed for quick relocation to optimize the lighting (i.e. applying more light in areas that are not fully illuminated by the overhead fixtures) for the photography. Further equipment details are summarized in the right side of Figure 3.12.

Figure 3.12: Flow visualization setup and equipment.

Photography was done using a digital Canon PowerShot G3 featuring a 4 *MP* CCD sensor, a 4 $\times$ optical zoom, a 35 – 140 *mm* lens, and a $f/2.0 - 3.0$ aperture (Canon, 2002). Usually, a high end camera equipped with a macro lens would be required to produce detailed close up images, however, it was found that by using a mid range camera equipped with a good quality lens and manual focus and aperture options, good quality images can still be produced. Typical settings used were: an $f/5$ aperture, a 7*mm* focal length, and a five second exposure time. A tripod was used, with a two second timer delay to ensure a stable and well focused image capture. Scales were printed on bright coloured paper available from office supply stores to be used for quantitative location of flow features seen in the images. It was found that green paper worked best with the post-processing steps of the images. The following section provides a description of the post-processing steps.

3.6.2 Image Post-Processing

While the UV light sources used provided a substantial amount of UV light, some visible purple-blue light was also present. To remove this purple-blue light, and enhance the quality of the image, a post-processing step was done.

Colour digital images are composed of pixels, and in the Red Green Blue (RGB) colour standard, each of the pixels is composed of a combination of red, green, and blue. Most raster (i.e. pixel) based image processing software allow manipulation of each colour component in the entire image by grouping them into channels, for red, green, and blue. In this case, the aim was to remove the purple-blue, a colour mostly controlled by the blue channel. Thus, a simple removal of the blue channel yielded the required result. The Channel Mixer function in Adobe Photoshop Creative Suite 4 (CS4) was used to produce the post-processed images. An example is shown in Figure 3.13. Additional adjustments in the brightness and contrast controls were also done to enhance the flow visualization patterns.

Figure 3.13: Image post-processing done through colour channel manipulation.

3.7 Data Reduction Methods and Uncertainty

3.7.1 Primitive Flow Variables

The primitive flow variables, such as M , P_o , P , T , ρ , V , u , v , w , α , β , were obtained by reducing the seven-hole pressure probe data from the main measurement plane located at $1.4 C_x$ from the leading edge (see Figure 3.3). The calibration and data reduction procedures for the seven-hole probe data are described in detail in Appendix A. The primitive flow variables obtained are averaged and used to further quantify the flow using a range of flow coefficients. The averaging methods and the coefficients are presented in the following sections.

3.7.2 Flow Averaging

In the context of this work, flow averaging is used to reduce several measured values from measurement points along a line (along one blade pitch corresponding to a particular

spanwise position) to one representative value; this is referred to as pitchwise averaging. It is also used to reduce several values from several measurement points on a plane (points corresponding to the measurement grid) to one representative value; this is referred to as planewise averaging.

Flow averaging allows a non-uniform flow (a flow with different values for its variables at different spatial locations) to be represented by a single value for each of its variables. In turbomachinery flows, this allows for assessment of the flow (e.g. loss coefficients) to be based on single values representative of inlet and outlet conditions. Two common methods of flow averaging used in turbomachinery flows, are mass-averaging, and area-averaging:

Mass-averaging:

$$\text{pitchwise, } \phi' = \frac{1}{\dot{m}_{pitch}} \int_0^s \phi \rho u dy dz \quad (3.6a)$$

$$\text{planewise, } \phi'' = \frac{1}{\dot{m}} \int_0^{0.5h} \int_0^s \phi \rho u dy dz \quad (3.6b)$$

Area-averaging:

$$\text{pitchwise, } \phi^* = \frac{1}{s} \int_0^s \phi dy dz \quad (3.7a)$$

$$\text{planewise, } \phi^{**} = \frac{1}{A} \int_0^{0.5h} \int_0^s \phi dy dz \quad (3.7b)$$

where, ϕ is the variable to be averaged.

Stagnation pressure is related to entropy (see next section); an extensive property, i.e. a property that is proportional to the mass in the system. Thus, in this work, mass-averaging is used for averaging stagnation pressure values.

For static pressure, Cumpsty & Horlock (2006) argued that an appropriate averaging method is derived from the definition of static pressure, i.e. a force per unit area, and thus a justification exists for area-averaging static pressure values. The author of the present work agrees, and area-averaging is used for static pressure values.

For velocity values, area-averaging allows for continuity to be satisfied, and thus velocity values are area-averaged in this work.

3.7.3 Losses

Loss can be quantified in terms of entropy production. Flow through the test cascade is assumed to be adiabatic (ΔT_o between blades and endwall of less than 0.4°C , Jouini, 2000), and air is approximated as an ideal gas. Therefore, in the absence of heat transfer, the only factor contributing to a change in entropy is flow irreversibilities. For adiabatic flow of an ideal gas through a stationary blade row, stagnation temperature is constant, and therefore the change in entropy depends only on the stagnation pressures as given by:

$$\Delta s = -R \ln \left(P_{o2}/P_{o1} \right) \quad (3.8)$$

or, for small changes in stagnation pressure (e.g. cascade flows)

$$\Delta s = -R \left(\Delta P_o / P_{o1} \right) \quad (3.9)$$

Thus for cascade flows, loss of stagnation pressure is directly related to increase of entropy. Several loss coefficients have been defined in the literature and Denton (1993) provides a detailed review.

In this study, the stagnation pressure loss coefficient (Y) is used. The stagnation pressure loss coefficient is used by most turbine designers, and thus it was chosen for this study. It will be used in quantifying loss details in the flow; for contour plots, pitchwise plots, and integrated values.

The stagnation pressure loss coefficient was defined as:

$$Y_{total} = \frac{P''_{o1} - P''_{o2}}{P''_{o2} - P_2^{**}} \quad (3.10)$$

where the double primes indicate planewise mass-averaged values, and the double asterisks indicate planewise area-averaged values as defined in the previous section.

The stagnation pressure loss coefficient was also used for detailed examination of the flow loss; for contour plots, its definition was slightly modified:

$$Y_{local} = \frac{P_{o1,CL} - P_{o2}}{P''_{o2} - P_2^{**}} \quad (3.11)$$

the mass-averaged inlet stagnation pressure (P''_{o1}) was replaced by the inlet centre line stagnation pressure ($P_{o1,CL}$), while the mass-averaged outlet stagnation pressure (P''_{o2}) was replaced by the local outlet stagnation pressure (P_{o2}).

For pitchwise plots, the definition of Y was also modified as follows:

$$Y_{pitch} = \frac{P_{o1,CL} - P'_{o2}}{P''_{o2} - P_2^{**}} \quad (3.12)$$

where the single prime indicates a pitchwise mass-averaged value.

As discussed in the previous chapter, probe measurements were not collected closer than 2.5% span from the endwall due to wall interference effects. In calculating the overall integrated loss values, the contribution from this section was estimated by using the static pressure at 2.5% as the limiting stagnation pressure at the endwall. These values were also compared to the static pressure tap values from the endwall and showed good agreement since there was no significant spanwise static pressure gradient near the endwalls at the measurement location of 1.4 C_x .

3.7.4 Mixed-out Loss

The measured losses defined in the previous section refer to losses that occur between the inlet boundary layer traverse plane ($2.0 C_x$ upstream) and the downstream measurement plane ($1.4 C_x$ downstream). The flow at the measurement plane is spatially non-uniform and additional losses will be generated as the flow mixes to a uniform condition. The losses at this condition, where the flow is spatially uniform, are referred to as the mixed-out losses. Mixed-out losses quantify the loss potential of the measured flow, by providing estimates of the additional losses that could be generated downstream. Additionally, they allow for comparison of data between different facilities, where the flow measurement planes may have been located at different distances.

The mixed-out losses were calculated by employing a control volume analysis, applying the continuity, momentum, and energy equations. The control volume encompassed one blade pitch, spanning from the endwall to midspan, starting from the measurement plane, to a hypothetical fully mixed-out plane. The analysis assumed constant area mixing, while neglecting shear forces. Thus, the calculation does not account for additional endwall loss production downstream of the measurement plane. The details of the mixed-out loss calculations are given in Appendix D. The definition of the mixed-out loss coefficient is given here:

$$Y_{mixed-out} = \frac{P_{o1}'' - P_{o,mix}}{P_{o,mix} - P_{mix}} \quad (3.13)$$

The definition of the mixed-out loss coefficient is similar to the measured loss coefficient, except that the averaged quantities at the measurement plane (subscript 2) are replaced by the quantities at the mixed-out plane (subscript *mix*) as derived in Appendix D.

3.7.5 Loss-Breakdown Scheme

The profile and the secondary losses were evaluated using the loss break-down used in the Kacker & Okapuu (1982) loss system. In this method, the profile losses are obtained from the results at midspan where the influence of the secondary flow on the blade surface flow is small. The secondary losses would then be obtained by subtracting the profile losses from the overall losses. This loss break down scheme was applied to the stagnation pressure loss coefficient (Y). The secondary loss flow coefficient is calculated as:

$$Y_{secondary} = Y_{total} - Y_{profile} \quad (3.14)$$

where

$$Y_{profile} = \frac{P_{o1,CL} - P'_{o2,ms}}{P''_{o2} - P_2^{**}} \quad (3.15)$$

This scheme assumes that profile losses are constant along the span, and equal to the midspan losses.

3.7.6 Exit Flow Angle Deviation, Streamwise Vorticity, and Secondary Kinetic Energy

In order to compare and understand the secondary flow fields downstream of the cascade, several flow parameters were investigated. These parameters, namely, the streamwise vorticity coefficient, and the secondary kinetic energy coefficient are based on velocity components normal to the primary flow direction. Pitchwise area-averaging is employed on all velocity quantities when needed (indicated by a single asterisk).

Deviation from the exit flow angle (ψ) provides information about the level of over-turning or underturning. It is defined relative to the midspan exit flow angle as follows:

$$\psi = \beta^* - \beta_{ms}^* \quad (3.16)$$

where

$$\beta_{ms}^* = \tan^{-1} \left(\frac{v_{ms}^*}{u_{ms}^*} \right) \quad (3.17a)$$

$$\beta^* = \tan^{-1} \left(\frac{v^*}{u^*} \right) \quad (3.17b)$$

Vorticity (ω) is a vector, defined as the curl of the velocity vector ($\vec{\nabla} \times \vec{V}$), expanded as follows:

$$\omega_x = \frac{\partial w}{\partial y} - \frac{\partial v}{\partial z} \quad (3.18a)$$

$$\omega_y = \frac{\partial u}{\partial z} - \frac{\partial w}{\partial x} \quad (3.18b)$$

$$\omega_z = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \quad (3.18c)$$

Streamwise vorticity is the vorticity present in a plane normal to the direction of the mainstream flow, and can be calculated from the components of the vorticity vector as defined below:

$$C_{\omega_s} = \frac{c}{V_{2,is}} (\omega_x \cos \beta_{ms}^* + \omega_y \sin \beta_{ms}^*) \quad (3.19)$$

Calculation of ω_y would require measurements at closely spaced downstream planes to obtain the velocity gradient in the axial direction ($\partial w / \partial x$). However, using the method outlined in Perdichizzi (1990), based on the steady inviscid Euler equations and assuming constant total enthalpy and ideal gas, ω_y can be estimated from a single measurement plane by:

$$\omega_y = \frac{1}{u} \left[v\omega_x + \frac{a^2}{\gamma} \frac{\partial (\ln P_{o2})}{\partial z} \right] \quad (3.20)$$

Yaras (1990) investigated the validity of the inviscid flow assumption in incompressible flow by comparing the single plane inviscid flow ω_y estimate to values calculated from two closely spaced planes. He found very good agreement between the two methods everywhere, except in areas where the velocity gradients were changing most rapidly (i.e. areas where viscosity effects are strongest), with a maximum local error of 10% between the two methods.

Secondary kinetic energy is defined as the kinetic energy associated with the secondary velocity components. The coefficient of secondary kinetic energy used was:

$$C_{SKE} = \frac{\rho(v_{sec}^2 + w_{sec}^2)}{\gamma P_{2,CL} M_{2,is}^2} \quad (3.21)$$

where

$$v_{sec} = v \cos \beta_{ms}^* - u \sin \beta_{ms}^* \quad (3.22a)$$

$$w_{sec} = w \quad (3.22b)$$

This coefficient was used in contour plots, but was also mass-averaged over the entire measurement plane to provide overall results.

3.7.7 Uncertainties in Measured Values

To quantify the uncertainties in the measured values, the single-sample method of Moffat (1982) was followed. In this method, experimental errors are classified into one of two categories: bias errors, or random errors. Bias errors are systematic and consistent errors, errors that result in a constant offset from the true value. For example, a poorly calibrated pressure transducer can introduce a bias error. Bias errors are minimized through well established instrument calibration procedures and overall wind tunnel operating procedures passed down from researcher to researcher. Random errors on the other hand, are errors

that arise from random fluctuations in the measurements. The effects of shear flow, turbulence and Reynolds number on the seven hole probe measurements can introduce both random and bias errors. The influence of these effects on the uncertainties in measured values are described in Johansen *et al.* (2001), and more specifically for this wind tunnel in Corriveau (2005).

While the author's uncertainty analysis follows the long-established methodology of Moffat (1982), it is worth noting that more recent methods (see Taylor, 1997) further classify bias errors as either fixed errors, or variable deterministic errors. Fixed errors are defined as repeatable bias errors, e.g. a poor calibration that results in a consistent offset from the true value. Variable deterministic errors are bias errors that may *appear* to be random, but in fact are not. An example could be changes in the atmospheric pressure, resulting in a variable, but deterministic offset that can be corrected for by tracking the changes in the atmospheric pressure.

Using statistical analysis, the uncertainty of a measured value can be described as follows:

$$\delta X = \pm 2\sigma \tag{3.23}$$

where σ is the standard deviation of the data; and if the data follows a Normal distribution (typical of data with random error), then δX represents the range where the true value lies with a 95.4% confidence level. In other words, the experimenter is 95.4% certain that X is between $X + \delta X$ and $X - \delta X$, traditionally an acceptable confidence level.

As per the method of Moffat (1982), a measured quantity, X , is a function of several variables each contributing a source of error:

$$X = f(x_1, x_2, \dots, x_{n-1}, x_n) \quad (3.24)$$

where n is the total number of variables (i.e., error sources). The effect of each variable on X can be quantified by:

$$\delta X_i = \left(\frac{\partial X}{\partial x_i} \delta x_i \right) \quad (3.25)$$

where δx_i is the uncertainty of each variable, and $\frac{\partial X}{\partial x_i}$ represents the effect of each variable on the measured quantity X . If the relationship between the variable and the measured quantity is explicitly known then this partial derivative can be evaluated directly; otherwise, it can be obtained by independently perturbing each variable (within its uncertainty) to determine the effect on X . The root sum square of all the effects ($\delta X_1 \dots \delta X_n$) can be then used to determine the uncertainty of the measured quantity:

$$\delta X = \pm \sqrt{\sum_{i=1}^n \left(\frac{\partial X}{\partial x_i} \delta x_i \right)^2} \quad (3.26)$$

For example, the uncertainty in the pressure measurements depends on the uncertainty in the pressure transducer calibration and the fluctuation in the measured pressure signal. The transducer voltage is converted to pressure using the following equation:

$$P = mV + z \quad (3.27)$$

where P is the pressure, m is the transducer calibration curve slope, V is the measured transducer voltage, and z is the voltage value at zero pressure (i.e. the y-intercept of the line). The transducers used for measurements in this experiment were discussed in

Section 3.3.3 and the transducer calibration curves are given in Appendix B. Using Equation 3.26, the uncertainty in the pressure measurement can be determined by the following equation:

$$\delta P = \sqrt{\left(\frac{\partial P}{\partial m}\right)^2 \delta m^2 + \left(\frac{\partial P}{\partial V}\right)^2 \delta V^2 + \left(\frac{\partial P}{\partial z}\right)^2 \delta z^2} \quad (3.28)$$

by differentiating Equation 3.27, the uncertainty in the pressure becomes:

$$\delta P = \sqrt{(V\delta m)^2 + (m\delta V)^2 + \delta z^2} \quad (3.29)$$

where δm is the uncertainty in the calibration curve slope, δV is the uncertainty in the voltage, and δz is the uncertainty in the zero pressure voltage of the calibration curve. δm , and δz can be obtained from the linear regression used to obtain the calibration curve equation for each transducer. Jeffries (2000) found that the uncertainty on the voltage signal (δV) at the frequencies and sampling times used in this study (2000 Hz and 0.1 s) was about $\pm 0.25\%$ within a 95.4% confidence level.

Table 3.2: Uncertainty values.

Pitchwise position, y	$\pm 0.2 \text{ mm}$
Spanwise position, z	$\pm 0.3 \text{ mm}$
Pressures, P, P_o	$\pm 0.2\%$
Mach number, M	± 0.01
Flow angles, α, β	$\pm 1^\circ$
Loss coefficient, Y	± 0.0015 to ± 0.0010 (for $M_{2,is}$ 0.69 to 0.89)

Corriveau (2005) presents a complete analysis of the uncertainty for measurements in this wind tunnel using the above described method. His analysis remains valid for this present study, and the measured values and associated uncertainties are summarized in Table 3.2. Additionally, a repeatability study performed on the baseline flat endwall, based on a complete tear down of the wind tunnel by two independent operators was

performed; one operator being the present author, the other was Taremi (2011). There was good agreement found between the two, well within the uncertainty values presented in Table 3.2.

To ensure the experimental measurements are within the expected uncertainty, and detect onsets of any problems, the transducer zero pressure voltage was checked daily for each transducer. Further, seven-hole probe pressure traverses were routinely complemented by Kiel pressure probe traverses showing good agreement.

Computational Method

4.1 Introduction

This chapter provides a description of the computational fluid dynamics (CFD) simulations that were carried out. A summary of the solver used and a description of the computational domain are first provided. The grid generation, and convergence criteria are then presented with some comparisons to experimental data for validation.

4.2 Simulation Cases and Computational Domain

Complementary CFD simulations at the design outlet Mach number of 0.80 were performed for all three cases: the baseline flat endwall, the cavity endwall, and the contoured endwall. The simulations complement the cascade experimental measurements by modelling the experiment as much as possible, with the aim of providing further insights into the flow physics by providing flow data from areas not accessible in the experiment (e.g. inner passage flow field data).

The simulations were carried out using the ANSYS CFX 12 commercial software package, a 3D Reynolds-Averaged Navier-Stokes (RANS) solver. The Shear-Stress-Transport

Blade Surface: No-slip adiabatic wall.

Endwall: No-slip adiabatic wall.

Midspan Plane: Symmetry.

Pitchwise Boundaries: Translational periodic.

Inlet: Total pressure and turbulence profile, with flow direction and constant total temperature.

Outlet: Area-averaged static pressure.

Dimensions: $2.0 C_x$, $3.5 C_x$, s .

Coordinate axes: x , y .

The computational domain closely models the experiment, and is shown in Figure 4.1 with the boundary conditions imposed. The cascade was modelled for one pitchwise pas-

sage, with one half-span blade, using periodic and symmetry boundary conditions. The domain inlet was located at $2.0 C_x$ upstream of the leading edge, while the outlet was at $3.5 C_x$ downstream of the leading edge. Inlet total pressure profile obtained from the experiment (as measured at $2.0 C_x$) was used for the inlet boundary condition with air modelled as an ideal gas at a constant total temperature of 285 K, appropriate assumptions to simulate the conditions of the high speed wind tunnel experiment. Profiles of turbulence kinetic energy (k), and specific turbulence dissipation rate (ω) were also used to define the turbulence boundary conditions. These profiles were obtained by simulating the boundary layer growth upstream of the blade passage using a computational pre-domain and extracting the turbulence quantities at a plane where the computational boundary layer closely matched the experimentally measured boundary layer parameters. The inlet total pressure, and turbulence profiles were oriented at the design inlet flow angle ($\beta_1 = 43.5^\circ$) as the inlet boundary condition, and are given in Figure 4.2. At the outlet, an average static pressure of 85447.5 Pa based on the experimental isentropic Mach number of 0.80 was specified.

Figure 4.2: Inlet total pressure (measured values) and computed turbulence profiles.

The remaining boundaries, the bottom endwall, and the blade surface, were set to adiabatic no-slip walls. A symmetry boundary condition was applied at midspan plane, and translational periodic conditions were imposed on the pitchwise boundaries. Although for the flat with cavity, and the contoured experimental cases, the cascade is not symmetric about the midspan (top endwall is flat), a significant portion of the flow is two-dimensional and therefore the secondary flows of the top and bottom endwalls are believed to be largely independent.

4.3 Grid Refinement and Solution Convergence

A hexahedral structured grid was created using ANSYS ICEM CFD 12 for each of the three cases. In creating the grid, the two-dimensional geometry of the blade profile was first considered, and once a good quality grid was obtained, the grid was extruded in the spanwise direction to the endwall where it was further refined. An O-grid was employed around the blade, to allow for orthogonal elements, and element concentration around the blade profile. Typically, a grid was divided into a total of about 80 blocks, with a total of about 5.6 million nodes. A typical grid is shown in Figure 4.3.

Solution convergence was assessed by examining the normalized residuals as well as the calculated stagnation loss coefficient at the outlet (Y_{outlet}). Typically, maximum residual values of about 10^{-5} were achieved, with RMS residual values of about 10^{-7} . The stagnation pressure loss coefficient at the outlet reaches a constant value after about 500 iterations. A typical convergence history is shown in Figure 4.4.

When refining the grid, the recommended guidelines given by the ANSYS 12 user manual for a high-quality grid were followed. To ensure the validity of the Gamma-Theta transition model, y^+ values for the first element from the wall for all wall surfaces were kept below 1, with an expansion ratio to the following node of equal to or less than 1.1.

Figure 4.3: Typical hexahedral grid generated, flat endwall shown.

The recommended grid quality criteria and the quality achieved are summarized in Table 4.1. By matching the recommended quality criteria (e.g. volume change less than two), along with the y^+ and expansion ratio requirements of the transition model, the resultant grids were of very high quality. This was particularly true in the pitchwise and axial directions around the blade profile; increasing the node count there yielded little change in pressures at the trailing edge or downstream loss coefficients. Further, the computed blade loading distribution was in good agreement with the experiment as shown

in Figure 4.5. This suggested that the simulation is modelling the experiment conditions reasonably well.

Figure 4.4: Typical convergence history, flat endwall final grid shown.

Table 4.1: Summary of grid quality

Criterion	Recommended	Achieved
Aspect ratio	< 1000	< 850
Minimum angle	> 18	> 37
Volume change	< 2.0	< 1.9
Determinant	> 0.3	> 0.8
y^+ of first node	< 1	< 1
Expansion ratio	≤ 1.1	≤ 1.1

A grid independence study using the flat endwall was performed. Six grids were generated with the node counts ranging from 0.38 to 7.4 million nodes with the minimum grid quality criteria maintained (Table 4.1). Comparison of the predicted stagnation pressure loss coefficient at $1.4 C_x (Y)$ to the experiment for each grid is shown in Figure 4.6.

Figure 4.5: Typical comparison of computational and experimental midspan blade loading, contoured endwall shown.

Figure 4.6: Comparison of CFD grids to experiment in terms of stagnation pressure loss coefficient at $1.4 C_x$.

The chosen grid had 5.6 million nodes as highlighted in Figure 4.6. This grid provided a grid independent solution with very little value added in increasing the node count beyond

this as shown by the levelling of Y in the figure. The chosen grid also allowed for a reasonable computational time. The simulations were run on a 36 partition cluster, with run durations of about 10 hours.

The grid produced a Y value of 0.047, while the experiment Y value was 0.041 as highlighted in Figure 4.6. CFD (specifically a RANS solver employing a turbulence model) typically predicts higher losses, as it often lacks in its ability to model the mixing processes leading to inaccurate total pressure loss computations. Full comparison of the computational results to the experiment are presented within the next chapters.

Flat and Cavity Endwalls at Design Mach Number

5.1 Introduction

In actual engine turbines, a space exists between the rotor disk and the stator ring to provide the required mechanical clearance for the rotor disks to rotate. This space manifests itself as a cavity at the rotor-stator interface as previously shown in Figure 1.2. The effects of platform geometry and purge flow have been investigated at low speeds. However, very little research has been done at high speeds, i.e. at the compressible flow conditions, representative of high-pressure (HP) turbines.

In this chapter, results at design conditions for a baseline flat endwall are presented along with results from an endwall with an upstream cavity approximating the gap between the rotor and stator in actual engines. Both endwall geometries are combined with the same blade geometry (SL2P). The cavity on the endwall panel is an approximation to the engine cavity of the HP turbine stage of an IAE V2500 turbofan engine.

The inlet boundary layer, and the blade loading of both endwalls are first presented. A discussion of the inner passage flow field is then presented, with flow visualization results

of the endwall and suction surface from both endwalls. Computational results are also presented for both endwalls that provide insights into the flow physics that complement the interpretation of the flow visualization results.

Next, a discussion of the downstream measurement results is presented, including losses, secondary kinetic energy, and streamwise vorticity. Complimentary CFD results provide a comparison of the computations to the experiment at the downstream measurement plane.

This chapter aims to present an understanding of the baseline flat and cavity endwalls with regards to flow physics and losses through both experimental and computational results. In a similar style of presentation, in Chapter 6, the results from the flat and cavity endwall will be compared to a contoured endwall at design conditions, with the performance of the endwall contouring being assessed.

5.2 Inlet Endwall Boundary Layers

The upstream boundary layer was traversed at $2.0 C_x$ upstream of the leading edge of the cascade using a goose-neck pitot probe with a diameter of 0.9 mm. This was done at the design Mach number of 0.80 for both the flat, and the cavity endwalls. The inlet total pressure profile in the endwall boundary layer is required to determine the net loss production within the blade passage. The type of flow, be it laminar, transitional or turbulent, impacts the formation of the endwall flows downstream, and thus characterizing the boundary layer is important.

Figure 5.1 shows the boundary layer profile in terms of the inlet Mach number ratio, $M_1/M_{1,ms}$, where $M_{1,ms}$ is the midspan inlet Mach number. It was found that both endwalls had very similar inlet endwall boundary layers. The shape factor (H) of about 1.3 shown in Figure 5.1, indicates essentially a zero pressure gradient two-dimensional turbulent boundary layer for both cases.

Figure 5.1: Inlet endwall boundary layer comparison of baseline flat endwall and cavity endwall.

5.3 Midspan Blade Loading Distributions

The static pressure taps on the suction side and pressure side of the two middle blades at midspan (see Figure 3.3) allowed for measurement of the blade loading on the cascade for both endwalls. Blade loading measurements were collected for the design outlet Mach number of 0.80 using both endwalls. The blade profile (SL2P) was designed using CFD (PWA CFD predictions); the experimental design blade loading was matched to the computational blade loading by fine tuning the blowing pressure of the wind tunnel.

Figure 5.2 shows good agreement between the experiment and the PWA CFD prediction values for the same outlet Mach numbers, indicating a good match between the experiment and the design condition. It is also evident that the midspan blade loadings for both endwalls are virtually identical, suggesting that any differences in the blade passage flow due to the presence of the cavity did not noticeably influence the midspan loading.

Figure 5.2: Experimental and CFD (PWA design prediction) blade loading comparison for flat and cavity endwalls.

The author's own CFD post-diction blade loading results (shown in Figure 4.5) are virtually identical to the PWA CFD prediction blade loading results, and are thus not repeated here. However, detailed post-diction CFD results of the passage and downstream flow field are provided within Sections 5.4.3, and 5.5.4.

5.4 Inner-passage Flow Field Behaviour

5.4.1 Endwall Flow Visualization

To help in the interpretation of the flow physics, a fluorescent oil surface flow visualization was conducted on both endwalls; the baseline flat, and the cavity endwall. The flow visualization method was explained in detail in Section 3.6. The results from the baseline flat endwall case are first discussed, followed by those of the cavity endwall.

Considerable work has been carried out on the investigation of secondary flows on flat endwalls (e.g. Sieverding (1985), Hodson & R.G. (1987), and Wang *et al.* (1997)), that

has yielded several detailed descriptions of the endwall flow within the blade passage at design incidence, albeit mostly at low speed conditions. The author's observations and the description of the flow field presented here are in fairly good agreement with those in the literature.

Figure 5.3 shows the flow visualization results of the baseline flat endwall on the left, with the corresponding interpretation on the right. The flow is entering from the bottom of the figure and exiting from the top left.

Figure 5.3: Flat endwall flow visualization results, with interpreted separation lines on the right.

As the incoming flow approaches the leading edge of the blade along the stagnation streamline (dotted line), it encounters an adverse pressure gradient, along with the pressure gradient set up by the turning of the flow by the blade passage, the inlet boundary layer rolls up at the saddle point labelled A_1 to form the horse-shoe vortex. Two major separation lines associated with the formation of the horse-shoe vortex are present. The first separation line S_1 , marks the separation of the inlet boundary layer from the endwall, where the boundary layer fluid is directed to either side of the blade, on the pressure side S_{1P} , and the suction side S_{1S} . The second separation line S_2 , marks the lift-off of the horse-shoe vortex, with the pressure side leg at S_{2P} , and the suction side leg at S_{2S} . The

separation is quite strong, as evident from the strong convergence of the surface shear stress vectors shown in the flow visualization.

The separation lines S_{1S} and S_{2S} converge as they round the suction side of the leading edge. At this point, it is believed that the cross-passage flow drives the suction side of the horse-shoe vortex off of the endwall. Meanwhile, the separation lines S_{1P} , and S_{2P} , related to the pressure-side leg of the horse-shoe vortex, extend across the blade passage, where the pressure side leg of the horse-shoe vortex at S_{2P} , is believed to evolve into the passage vortex as it is fed by boundary layer fluid from the endwall.

Another major separation line that is evident in the flow visualization, is marked as S_3 , and seems to originate at a point close to the intersection of the S_{1S}/S_{2S} separation lines, and the blade suction surface. It is associated with the formation of the corner vortex, which is thought to be formed as a result of the strong cross-passage flows on the endwall meeting the blade suction surface.

There is very little in the open literature on experimental flow visualization of a cavity ahead of a turbine blade cascade. de la Rosa Blanco *et al.* (2005) presented some flow visualization of a turbine blade cascade with a backwards facing step. However their flow visualization results were not very clear. More recently, MacIsaac & Sjolander (2013) presented computational results of a low-speed turbine cascade with a cavity (similar to the one in this study) with CFD flow visualization results that seem to agree with the interpretations observed in this study. The author's own computations support the flow interpretations provided, with some CFD results shown here to aid in the understanding of the flow physics.

Figure 5.4 shows the flow visualization results of the cavity endwall on the top, with the corresponding interpretation on the bottom. The edge of the cavity is marked with a solid blue line. The cavity is located parallel to the leading edge plane, such that the inlet flow direction approaches the cavity upstream of the passage with a relative angle.

Figure 5.4: Cavity endwall flow visualization results with interpretation.

It is evident from the flow visualization that the flow at the endwall is significantly altered by the introduction of the cavity upstream. The incoming flow approaches the edge of the cavity with a relative angle and thus it has a pitchwise momentum component in the suction-to-pressure-side direction. As it approaches the edge of the step, it encounters an adverse pressure gradient, where it is believed to be stronger ahead of the leading edge stagnation point of the blade (due the combined effect of both the cavity and the blade), and weaker ahead of the entry into the passage. It is believed that this results in a separation of the inlet boundary layer at the separation line marked by the dashed line $S_{C,1}$. This three-dimensional separation is thought to form a recirculation zone within the cavity, a “cavity vortex”, made up of inlet boundary layer fluid, which then reattaches at the reattachment line marked by the dotted line R_C . Figure 5.5 provides several cross-sectional views along the pitchwise direction, showing axial velocity streamlines of this “cavity vortex”, with the separation and reattachment points marked.

To help in understanding the flow physics computational fluid dynamics (CFD) is used. Figure 5.6 illustrates the formation of the cavity vortex, where a “suction side leg” of the cavity vortex is represented by the green stream-ribbons, and a “pressure side leg” of the cavity vortex is represented by the red stream-ribbons.

Since this inlet boundary layer has pitchwise momentum, in the region ahead of the blade, (where the axial adverse pressure gradient is highest, and the cross-passage pressure gradient is weakest) some of the recirculating flow travels in the pitchwise suction-to-pressure-side direction forming the “suction side leg” of the cavity vortex as shown by the surface shear stress vectors in Figure 5.4 and illustrated by the green stream-ribbons in Figure 5.6. As this flow travels further pitchwise to the middle of the adjacent passage where the adverse pressure gradient is lower, it escapes the cavity into the adjacent blade passage and is then pushed towards the blade suction side by the cross-passage pressure gradient along the second separation line $S_{C,2S}$ shown in Figure 5.4.

Figure 5.5: Cross-sectional views of the cavity vortex, at planes normal to the cavity.

Figure 5.6: Computational results showing the formation of the cavity vortex.

On the other hand, the separated flow closer to the middle of the passage, where the cross-passage pressure gradient is higher, can not overcome the cross-passage pressure gradient and is driven in the pitchwise pressure-to-suction direction as indicated by the surface shear stress vectors on the bottom of Figure 5.4 and illustrated by the red stream-ribbons in Figure 5.6, thus forming the “pressure side leg” of the cavity vortex, where it escapes into the passage along the separation line $S_{C,2P}$ shown in Figure 5.4.

Lastly, the separation line associated with the formation of the corner vortex is marked as S_3 on Figure 5.4. It appears to be a stronger separation line, and remains closer to the suction surface than that of the flat endwall case. This suggests that the cavity endwall could have a stronger corner vortex developing downstream. The author believes that the slightly higher momentum boundary layer fluid (fluid that is not separated at the cavity) exhibits the classical horse-shoe vortex separation, but at a saddle point further away from the endwall than that of the flat case, and thus not visible on the endwall but evidenced on the suction side surface flow visualization (shown in the next section).

The recirculated flow that escapes from the cavity vortex and turns into the blade passage is thought to mix with the classic pressure side leg of the horse-shoe vortex (forming above the endwall surface), and thus resulting in a stronger and larger passage vortex downstream. Thus, the passage vortex that will develop for the cavity endwall will be comprised of fluid from the recirculation zone within the cavity, and endwall boundary layer fluid from the blade passage. This composition is believed to lead to a significantly larger and stronger passage vortex for the cavity endwall than that of the flat endwall, which is consistent with the downstream results presented in Section 5.5. In section 5.4.3, the computational inner-passage results are further discussed, and show the formation of the horse-shoe vortex and its interaction with the cavity vortex.

Although the observed endwall flows are believed to be mostly steady, there is some evidence of unsteadiness in the surface flow visualization. Two regions that appear to

indicate some unsteadiness are highlighted in the top of Figure 5.4; the smeared limiting stream lines shown in the large square, and the zig-zags in the bifurcation shown in the small square. It is possible that the escaping of vortical fluid is unsteady, and that certain flow conditions can cause acoustic resonances in the cavity vortex significantly changing the behaviour of the escaped vortical fluid.

5.4.2 Suction Surface Flow Visualization

Figure 5.7 shows the suction surface flow visualization results, flat endwall on the left, and cavity endwall on the right. The view for both is oriented with the leading edge at the bottom of the figure, with a scale of surface fractional distance wrapped on the right, and span fraction at the trailing edge. Examining the flat endwall on the left, two dominant separation lines are indicated, marked S_4 and S_{2S} . S_4 is typically associated with the passage vortex; it emanates at roughly $0.50 \frac{S_s}{S_{s,max}}$, and indicates the penetration of the passage vortex along the span to be at $0.1 \frac{z}{h}$ at the passage outlet. S_{2S} is the extension of the suction side leg of the horse-shoe vortex that has lifted up from the endwall. Wang *et al.* (1997), have shown that the suction side leg can be lifted off the suction surface by the passage vortex and wrapped around it in a planet-sun configuration downstream. This is supported by the CFD results shown in the next section.

Figure 5.7: Comparison of flat and cavity endwalls; suction surface flow visualization with interpretation.

The right hand side of Figure 5.7 shows the suction surface flow visualization results for the cavity endwall. Again, the two dominant separation lines are indicated, marked S_4 (associated with the passage vortex), and S_{2S} (associated with the suction side leg of the horse-shoe vortex).

S_{2S} emanates at $0.10 \frac{S_s}{S_{s,max}}$, significantly earlier than that of the flat endwall. S_4 emanates at $0.40 \frac{S_s}{S_{s,max}}$, earlier than the flat endwall, and shows a slight increase of the penetration depth (roughly at $0.125 \frac{z}{h}$, a 0.025 increase) suggesting that stronger secondary flows are present. The more “smeared” oil-dye on the suction surface for the cavity endwall also suggests that the secondary flows are significantly stronger, and might possibly impact the profile flows at midspan. However, the midspan blade loadings presented earlier (see Section 5.3) do not appear to reflect the apparent differences in the blade surface shear stresses observed.

5.4.3 Computational Flow Physics and Visualization

Complementary CFD simulations at the design outlet Mach number of 0.80 were performed for both the flat and cavity endwalls. The simulations complement the cascade experimental measurements by modelling the experiment as closely as possible; a complete description of the computational method was provided in Chapter 4. In this section, the inner-passage computational results are examined to help in the understanding of the flow physics, while Section 5.5.4 presents the downstream computational results.

Figure 5.8 shows the computational results for the inner-passage flow field for the flat endwall. The flow physics are in good agreement with that observed in the experiment flow visualization. The left side of the figure, shows the classic formation of the horse-shoe vortex as the inlet boundary layer fluid separates at the indicated saddle point. The right side of the figure highlights the pressure side leg of the horse-shoe vortex, and the suction

Figure 5.8: Computational flow visualization for the flat endwall.

side leg of the horse-shoe vortex. As the pressure side leg traverses across the passage from the pressure side of the blade to the suction side of the blade, it picks up endwall boundary layer fluid and develops into the passage vortex downstream (marked in red). The suction side leg of the horse-shoe vortex develops into the counter vortex (marked in green), and has an opposite sense of rotation to that of the passage vortex. The computations showed that the suction side leg interacts with the suction surface boundary layer fluid, and later downstream is lifted off the suction surface by the passage vortex and is wrapped around it in a planet-sun configuration. Such a behaviour was also observed experimentally by Wang *et al.* (1997) using smoke wire visualizations, and is in agreement with the suction surface flow visualization results previously presented.

Figure 5.9 shows the computational results for the inner-passage flow field for the cavity endwall. The flow physics supports the observations made in the experiment flow visualization. The computations show that the low momentum inlet boundary layer fluid (shown in orange) with its pitchwise momentum in the pressure-to-suction side direction (due to the relative angle that the flow approaches the cavity) separates at the cavity

Figure 5.9: Computational flow visualization for the cavity endwall.

edge, and a strong recirculation zone is formed within the cavity. From this recirculation, the cavity vortex is formed at a saddle point as discussed earlier in Section 5.4.1. On the pressure side of this saddle point, flow from the cavity vortex is driven out of the cavity into the passage (due the drop in adverse pressure away from the blade leading edge), and is channelled towards the suction surface of the blade by the cross-passage pressure gradient. On the suction side, the passage vortex is kept within the cavity by the blade potential, and due to its pitchwise momentum traverses in the pitchwise suction-to-pressure side direction until it reaches the adjacent passage where it is allowed to escape from the cavity into the passage and is then channelled back towards the suction surface of the blade by the cross-passage pressure gradient.

Further from the endwall, the higher momentum inlet boundary layer fluid (shown in yellow) exhibits the classic horse-shoe vortex separation as highlighted in the enlargement in the box in Figure 5.9. The separation occurs at a saddle point higher off the endwall than in the flat case, but still with the typical pressure side leg and suction side leg of the horse-shoe vortex. The pressure side leg is enlarged by the high loss fluid originating from the separated cavity vortical flow, leading to a stronger and larger passage vortex downstream, as shown in the figure. Meanwhile, the suction side leg continues downstream and develops into the counter vortex rotating oppositely to the passage vortex.

The CFD visualization results support the interpretation of the experimental flow visualization, but with some differences. In the experiment, flow from the cavity vortex escapes the cavity with a very curved trajectory towards the blade suction surface, as marked by the $S_{C,2S}$, and $S_{C,2P}$ lines on Figure 5.4. The CFD results on the other hand, show a weaker turn of this flow, and a lifting-off from the endwall away from the blade, closer to mid-passage. This sharp turning of the escaped flow while staying near the endwall observed only in the experiment is believed to contribute to the thickening of the near-endwall vorticity layer as will be shown in the experimental downstream results in Section 5.5.2. On the other hand, the lack of this behaviour in the CFD results might contribute to the CFD not showing any thickening of the near-endwall vorticity downstream of the passage as will also be shown in Section 5.5.4.

As highlighted by the blue stream-ribbons in Figure 5.9 (within the zoomed-in box), the computations suggest that fluid from the suction side leg of the horse-shoe vortex can instead become part of the cavity vortex, and end up being part of the passage vortex downstream. The end result could be a slight decrease in the size of the counter vortex downstream for the cavity endwall case. Some evidence of this has been observed in both the experimental and computational downstream results, which are presented next.

5.5 Downstream Flow Field Behaviour

5.5.1 Loss Distributions

Figure 5.10: Comparison of baseline flat endwall and cavity endwall: (a) Downstream local stagnation pressure losses. (b) Pitchwise averaged stagnation pressure losses.

Seven-hole pressure probe measurements were made at $1.4 C_x$ from the leading edge (see Figure 3.3) for both the baseline flat endwall, and the cavity endwall at the design Mach number of 0.80. Pressure losses, in the form of the local stagnation pressure loss coefficient (Y_{local}), and the pitchwise averaged stagnation pressure loss coefficient (Y_{pitch}) are presented in Figure 5.10.

Figure 5.10 (a) shows Y_{local} contour flood results for the flat endwall and the cavity endwall. The flat endwall is on the left, and cavity endwall is on the right. The view is of the measurement plane viewed looking upstream with the passage pressure side on the left, and the passage suction side on the right; both contour plots have the same scale for direct comparison. Looking at the flat endwall, the main secondary flow loss region is observed, with the maximum loss of 0.35 occurring at about 20% span from the endwall. A second smaller loss region at about 2.5% span is partially captured at the last traverse line near the endwall with a maximum loss of 0.26.

Comparing the flat endwall to the cavity endwall, the right side of Figure 5.10 (a) shows significant change in the flow structures, and a redistribution of the losses in the secondary flow region. The cavity significantly alters the secondary loss distributions, with the main loss region expanding significantly. Comparing the flat and cavity endwall loss regions, the cavity endwall has a larger loss region with local maximum loss values of 0.32 and 0.14 appearing in the now much larger overall main loss core. The second smaller loss region at about 2.5% span has increased in both size and intensity as a result of the presence of the cavity, with a maximum loss value of 0.32. In the next section it will be shown that these loss regions in both the flat and cavity endwall are associated with the vortical structures that develop in the passage.

Figure 5.10 (b) presents the pitchwise averaged pressure loss coefficient across the span for both the flat endwall, and the cavity endwall cases. Such plots are commonly used to compare the effects of geometric and aerodynamic modifications that affect the three-dimensional aspects of the flow, particularly the secondary flow near the endwall. Similar plots for other parameters are presented later. However, these plots have shortcomings and should be treated with some caution.

The pitchwise stagnation pressure loss coefficient was previously presented in Section 3.7.3, but its definition is repeated here:

$$Y_{pitch} = \frac{P_{o1,CL} - P'_{o2,y}}{q''_{2,y}} \quad (5.1)$$

where,

$P_{o1,CL}$ is the reference total pressure, measured on the centreline upstream of the cascade.

$P'_{o2,y}$ is the mass-averaged total pressure at the measurement plane at spanwise location y .

$q''_{2,y}$ is the mass-averaged dynamic pressure ($P''_{o2} - P_2^{**}$), at the measurement plane at spanwise location y .

Two shortcomings are identified that can cause the plot of Y_{pitch} to give a distorted picture of the losses:

- (i) The endwall is a region of both higher losses and reduced mass flow rate, and thus lower dynamic pressure. Therefore, the Y_{pitch} value is higher there because of both, the higher losses *and* the lower value of the dynamic pressure used to non-dimensionalize the losses. It could be argued that a truer picture of the losses would be obtained if a constant value of the dynamic pressure, such as the outlet value on the centreline, were used for the non-dimesionalization. This would have the effect of reducing the values of the loss coefficients shown on Figure 5.10 (b) in the endwall region. However, that is not the standard form for this plot.
- (ii) The second shortcoming arises from the use of the inlet centreline total pressure as the reference value. As discussed earlier, the secondary flow measured in the outlet plane is deficient in total pressure for two reasons: because of the losses generated inside the blade passage and because of the deficiency of total pressure already present in the endwall boundary layer upstream of the blade row (this issue also affects the contour plots shown in Figure 5.10 (a)). Thus, when a core of high loss is identified it is not known whether the losses are high because of loss generated inside the blade passage or because this is where inlet boundary layer fluid has ended up.

With the above considerations in mind, several observations from Figure 5.10 (b) can be made. The most notable area of loss increase occurs between 20% to 5%, span which corresponds to the main loss core that is significantly enlarged by the presence of the cavity. However, the largest differences occur near the endwall at 2.5% where the cavity is thought to contribute to the thickening of the near-endwall vorticity and considerably increasing the losses associated with the corner vortex (see Sections 5.4.1 and 5.4.3 for an explanation of the flow physics). It is suspected that even closer to the endwall, below 2.5% (where measurements could not be obtained), this trend may continue with the cavity having a significant impact on the losses in the near endwall region. The pressure losses presented here can be related to the downstream vorticity field presented in the next section.

5.5.2 Streamwise Vorticity Field

The development of the vortical flow structures within the passage was previously discussed in Section 5.4. To understand the downstream flow structures, the streamwise vorticity (C_{ω_s}) can be used to identify the vortical structures and quantify their strength and sense of rotation. C_{ω_s} was calculated from seven-hole pressure probe measurements (see Section 3.7.6) that were made at $1.4 C_x$ from the leading edge, for both the baseline flat endwall, and the cavity endwall at the design Mach number of 0.80. With the right hand coordinate system, positive vorticity corresponds to counter clockwise rotation, and negative corresponds to clockwise rotation when viewed from downstream.

Figure 5.11 (a) shows a colour flood of C_{ω_s} together with contour lines of the local stagnation pressure loss coefficient (Y_{local}), for both cases. Beginning with the flat endwall, it is seen that there are three vortical structures. The counter clockwise rotating passage vortex (developed from the pressure side leg of the horse-shoe vortex, see Section 5.4.3) and the clockwise rotating counter vortex (developed from the suction side leg of the horse-shoe vortex, see Section 5.4.3) are the dominant vortical structures. The clockwise

rotating corner vortex (developed as a result of the pressure to suction side cross flows meeting the suction surface of the blade) is barely visible at the lowest spanwise traversing position.

Figure 5.11: Comparison of baseline flat endwall and cavity endwall: (a) Streamwise vorticity field with stagnation pressure loss contour lines. (b) Pitchwise averaged exit flow angle deviation.

From the Y_{local} contour lines, the maximum loss core value of 0.35 appears to coincide with the counter vortex, with some elevated loss levels associated with regions where the counter and passage vortices interact with each other. This can be attributed to

the stronger mixing, and higher shear levels that are present in the region between the oppositely rotating vortices. Previous studies (e.g. Benner, 2003), have shown that for low-speed incompressible flows, most of the losses were associated with the passage vortex. More recently, in a related high-speed compressible flow study, Taremi (2011) has found similar results to those observed here, with the highest losses associated more with the counter vortex than the passage vortex. Lastly, the corner vortex appears to be associated with the endwall loss core as evident from the loss contour lines.

Comparing the flat endwall to the cavity endwall case, the large clockwise vorticity near the endwall spanning the entire pitch, as well as significant changes with the previously identified vortical structures are observed. The passage vortex has increased in size, as a result of being enlarged by the vortical flow from the cavity separation (i.e. cavity vortex). The corner vortex appears to have increased in strength significantly. Development of the cavity vortex was discussed in Section 5.4.1, where it was shown that it is comprised of a pressure side leg and a suction side leg as shown in Figure 5.6. It is believed positive vorticity of the escaped vortical fluid from the pressure side leg of the cavity vortex will be the most dominant (relative to the negative vorticity from the suction side leg of the cavity vortex). This dominantly positive vorticity interacts with the pressure side leg of the horse-vortex of the same sign, adding additional vorticity to it, and leading to the significantly enlarged passage vortex. In turn, this enlarged passage vortex gives rise to a stronger cross-passage shear flow resulting in an enlarged near-endwall negative vorticity, and a stronger corner vortex develops as stronger cross-flows meet the suction surface of the blade.

The enlarged passage vortex appears to be associated with the enlarged loss region as evident by the pressure loss contour lines. The much stronger corner vortex is also associated with the higher maximum (measured) loss core value of 0.32 near the endwall at about 2% span. The size of the counter vortex appears to have slightly diminished, this

may be attributed to the cavity vortex trapping some of the fluid from the suction side leg of the horse-shoe vortex and directing it to the pressure side leg (see Section 5.4.3).

Figure 5.11 (b) shows the pitchwise averaged exit flow angle deviation (ψ) (i.e. exit flow angle relative to midspan value). The cavity has a small effect on the level of underturning in the outer part of the endwall flow, causing a slight increase in underturning as can be seen from about 20% to 12% span. Below 12% span, ψ values for the cavity endwall exceed that of the flat endwall, and continue their increase, resulting in a significant increase to the level of overturning. This increase in overturning can be attributed to the reduction in streamwise momentum caused by the addition of low momentum fluid from the cavity upstream, which leads to more cross-flow in the passage flow under the action of the blade-to-blade pressure difference.

5.5.3 Secondary Kinetic Energy Field

To further clarify the flow physics and the production of secondary losses, the secondary kinetic energy coefficient (C_{SKE}) is also extracted from the downstream measurements. As discussed in Section 3.7.6, the secondary kinetic energy is kinetic energy associated with the secondary velocity components, and is an indication of eventual loss production through mixing. Figure 5.12 (a) shows a contour plot of C_{SKE} , with secondary velocity vectors, and Y_{local} contour lines overlaid. Figure 5.12 repeats the plot of the pitchwise averaged exit flow angle deviation (ψ) from Figure 5.11 (b) for easier reference.

Examining the flat endwall results on the left of Figure 5.12 (a), the largest secondary velocities and thus C_{SKE} values are shown to occur in the vortex-vortex interaction regions. The vortex-vortex interaction regions, are the region between the passage and counter vortex at about 15% span, and the region between the passage and corner vortex at about 3% span. The results from the cavity endwall on the right side of the figure, show much higher values of C_{SKE} in these same regions, spanning a larger area of the flow. This is

Figure 5.12: Comparison of baseline flat endwall and cavity endwall: (a) Secondary kinetic energy field with stagnation pressure loss contour lines and secondary velocity vectors. (b) Pitchwise averaged exit flow angle deviation.

the result of the significant increase in secondary kinetic energy due to the larger passage vortex, the near-endwall vorticity near the endwall, and the significant strengthening of the corner vortex as shown in the previous section. This increase in secondary kinetic energy (specifically from 10% span and below) due to the presence of the upstream cavity, is consistent with the significant increase in flow overturning as indicated by the flow angle deviation (ψ) previously presented, but repeated in Figure 5.12 (b).

Figure 5.13 (a), provides an alternative way to examine results in Figure 5.12 (a), with the secondary velocity vectors replaced with lines tangent to the secondary velocity

Figure 5.13: Comparison of flat endwall and cavity endwall: (a) Secondary kinetic energy field with stagnation pressure loss contour lines and secondary flow pseudo-streamlines. (b) Pitchwise averaged secondary kinetic energy.

vectors (pseudo-streamlines), highlighting that the regions with the highest secondary kinetic energy, are those between oppositely rotating vortices. Figure 5.13 (b) shows the pitchwise mass-averaged C'_{SKE} for both the flat and cavity endwalls. The presence of the cavity leads to a significant increase in secondary kinetic energy from about 20% span and below. An important loss production mechanism, as shown in previous low-speed studies (e.g. Benner, 2003), is the mixing out of the secondary flow structures and the resulting dissipation of their associated secondary kinetic energy. Mass-averaged secondary kinetic energy values (C''_{SKE}) of 0.0017, and 0.0058 were found for the flat endwall and the

cavity endwall cases respectively. Thus, the significantly higher secondary kinetic energy of the cavity endwall case indicates that the cavity endwall could generate higher losses downstream of $1.4 C_x$ than the flat endwall case, as will be discussed further in Section 5.6.

5.5.4 Computational Downstream Results

Complementary CFD simulations at the design outlet Mach number of 0.80 were performed for both the flat and cavity endwalls. A complete description of the computational method was provided in Chapter 4. The computational results describing the inner-passage flow field were perviously presented in Section 5.4.3, while the computational downstream results are presented here.

Figure 5.14 compares the pressure losses in terms of the local stagnation pressure loss coefficient (Y_{local}) between the experimental results (a) (repeated from Figure 5.10), and the computational results (b), at $1.4 C_x$. Noticeable differences (particularly for the flat endwall) in the magnitudes and locations of the peak losses were observed, with some broad similarities, particularly for the cavity endwall. Overall the computations exhibited larger and more confined loss cores, with higher maximum values than those observed in the experiment.

The flat endwall computations yield significant differences in the loss distributions to those of the experiment, with the secondary loss region being noticeably farther from the endwall than observed in the experiment. Further, both the main loss core (associated with the counter vortex), and the near endwall loss core (associated with the corner vortex), had much higher values of 0.45 and 0.39 than those of the experimental values of 0.35 and 0.26. The computations also showed the highest losses at the region closest to the endwall. In the experiment this region was outside the traverse region, and it is possible that the maximum loss core value associated with the corner vortex could peak below the lowest measurement location, and that even higher maxima occur closer to the endwall.

Figure 5.14: Comparison of downstream local stagnation pressure losses: (a) Experimental results. (b) Computational results.

Computations for the cavity endwall yielded broadly similar flow structures to those of the experiment, with the presence of the cavity leading to a second loss core at about 10% span as also seen in the experiment. However the computations predicted much higher maximum loss core values of 0.45 and 0.23 relative to the experiment values of 0.32 and 0.12. The loss core that is associated with the corner vortex at about 2.5% span is broadly similar in shape between the computation and the experiment, although the latter is only

partially captured. However, in the computations, the presence of the cavity appeared to reduce the losses in the very near wall region (maximum loss core value reduced from 0.39 to 0.35), while in the experiment it increased (maximum *measured* loss value increased from 0.26 to 0.32).

The higher losses and significantly different loss distributions, are attributed to the CFD's apparent inability to model (accurately) the diffusion and mixing processes leading to inaccurate total pressure loss predictions. Overall the computations seem to predict much higher losses than those seen in the experiment, with loss patterns that suggest further mixing out will occur (more so than for the experimental results), and thus the computational mixed-out losses are thought to be even higher when compared to the experimental mixed-out losses. The experimental and computational, measured losses, and mixed-out losses, will be presented in Section 5.6.

Figure 5.15 shows a comparison of the flow vorticity in terms of the streamwise vorticity coefficient (C_{ω_s}), and Y_{local} contour lines, between the experimental results (a), and the computational results (b), at $1.4 C_x$.

The computational results yielded significant differences in the location and strength of the three main vortical structures: the passage vortex, counter vortex, and corner vortex, and the loss cores typically associated with them. For the flat endwall case the computations predict a very large shift in the location of the passage vortex away from the wall. For the cavity endwall case, the counter vortex moves slightly towards the wall, while the passage vortex was shifted outward slightly. For both cases the computations predicted a smaller distance between the counter and passage vortices than that observed in the experiment.

Overall, larger and stronger passage and counter vortices are found in the computations. Similar to the experiment, the presence of the cavity in the computational results yields an increase in the size and strength of the passage vortex. However the thickening of the near-

Figure 5.15: Comparison of streamwise vorticity field with stagnation pressure loss contour lines: (a) Experimental results. (b) Computational results.

endwall vorticity and the strengthening of the corner vortex observed in the experiment do not appear to have been captured by the computations. This may be attributed to the different, and less drastic (than observed in the experiment) turning of the escaped cavity vortex fluid within the inner-passage predicted by the computations (see Figure 5.9 in Section 5.4.3).

As the loss production downstream will be partially dependent on the mixing out of these vortical flow structures and their associated secondary kinetic energy, the secondary kinetic energy coefficient (C_{ω_s}) becomes a useful parameter to indicate the total pressure losses yet to be generated downstream. Figure 5.16 shows a comparison of C_{ω_s} , with Y_{local} contour lines, and secondary flow pseudo-streamlines, between the experimental results (a) (repeated from Figure 5.13), and the computational results (b), at $1.4 C_x$. As was the case with the experiment, the computations show the largest C_{SKE} values in the vortex-vortex interaction regions. These are the regions between the passage vortex and the counter vortex, and between the passage vortex and corner vortex.

The presence of the cavity in the computations leads to a significant increase of secondary kinetic energy in the region between the passage vortex and the corner vortex as was the case in the experiment, with a computed planewise mass-averaged secondary kinetic energy value (C''_{SKE}) of 0.0030, and 0.0085 for the flat endwall, and the cavity endwall respectively. Overall, the computations predict much larger secondary kinetic energy values than those observed in the experiment. The computational and experimental secondary kinetic energy averaged values are further discussed along with the averaged measurement plane losses, and mixed-out losses in Section 5.6.

Figure 5.17 compares the experimental and computational pitchwise averaged results: (a) pitchwise averaged stagnation pressure losses (Y_{pitch}), (b) pitchwise averaged exit flow angle deviation (ψ), and (c) pitchwise averaged secondary kinetic energy coefficient (C'_{SKE}). The loss values from the computations are generally higher than those found in the experiment. Most notably, a very large discrepancy between the experiment and the computations exists for the midspan losses. This was somewhat unexpected. However, Taremi (2011) encountered a similar level of discrepancy in computations using the same SL2P airfoil geometry at the same conditions, but was able to slightly improve the agreement with the experiment midspan losses by running the simulations fully turbulent

Figure 5.16: Comparison of secondary kinetic energy field with stagnation pressure loss contour lines and secondary flow pseudo-streamlines: (a) Experimental results. (b) Computational results.

without the use of a transition model. The author of the present study, however, used the gamma-theta transition model (as mentioned earlier) to try to model the physics of the experiment as closely as possible.

For the flat baseline case, there was good agreement below 15% span. For the cavity endwall, the computations are consistently higher than the experiment until about 5% span, but they yielded a similar loss pattern to that seen in the experiment. The

Figure 5.17: Comparison of pitchwise averaged results: (a) Pitchwise averaged stagnation pressure losses. (b) Pitchwise averaged exit flow angle deviation. (c) Pitchwise averaged secondary kinetic energy.

computations predicted a much higher penetration depth of the secondary flow than that shown in the experiment. This is evident in Figure 5.17 (b), with the larger values of flow underturning at 20%, and in Figure 5.17 (c) with the higher secondary kinetic energy at 20% span. The computations gave somewhat similar trends, but the losses and secondary kinetic energy were significantly over-predicted.

5.6 Total and Mixed-out Losses

The planewise averaged (over the $1.4 C_x$ measurement plane) stagnation pressure losses (Y_{total}), the mixed-out stagnation pressure losses ($Y_{mixed-out}$), and the planewise averaged secondary kinetic energy coefficient (C''_{SKE}) are summarized in Figure 5.18. Y_{total} defined in Section 3.7.3 identifies only the losses generated inside the blade row, since the mass-averaged total pressure deficiency in the incoming flow has been subtracted out. $Y_{mixed-out}$ gives a value for the mixed-out losses as defined in Appendix D. C''_{SKE} is the mass-averaged secondary kinetic energy coefficient, and is related, but not directly comparable to the value obtained from a visual integration (i.e. area-averaging) of the pitchwise averaged values previously shown in Figure 5.13 (b). Experimental and computational results for both endwalls are given.

The experiment shows an 11% increase in losses (relative to the baseline flat endwall case) as a result of the cavity, with these losses increasing to 14% when mixed-out.

Figure 5.18: Experimental and computational losses, and secondary kinetic energy.

The computations did not show the same loss trends as the experiment. They predicted an 8.5% *decrease* in losses (relative to the baseline flat endwall case) for the cavity endwall case at 1.4 C_x , and almost no change in losses when mixed-out. This is due to the larger than expected (relative to the experiment) secondary flow region computed for the flat baseline case. MacIsaac & Sjolander (2013) carried out a similar, but low-speed study, comparing a flat endwall to a cavity endwall and obtained similar results. They found that the experiment yielded a 9% increase in total losses due to the cavity (relative to a flat baseline case), while the computations predicted a 6% decrease in total losses due to the cavity. This suggested that for complex flows involving such cavities, RANS type CFD might adequately predict the formation of the main vortical structures, but fails to model the mixing processes, leading to inaccurate total pressure loss predictions. For this case, the CFD predicted opposite loss trends to the experiment.

Experimental C''_{SKE} values of 0.0017 for the flat endwall and 0.0058 for the cavity endwall were found, an increase of about 240%. The secondary kinetic energy is kinetic energy associated with the secondary velocity components, and is an indication of eventual loss production from the dissipation and mixing out of the kinetic energy of the secondary flows. Thus, the higher experimental C''_{SKE} for the cavity endwall results is consistent with the larger secondary flows, and higher losses. If we were to assume that the dominant source of loss production is due to the dissipation and mixing out of the secondary kinetic energy, then the summation of the total losses (Y_{total}) and C''_{SKE} , should be close to the mixed-out loss ($Y_{mixed-out}$) value. However, the very small values of C''_{SKE} clearly demonstrate that this is not the case here. This suggests that in high speed flows, the dissipation and mixing out of secondary kinetic energy is not the dominant source of loss production, and that other sources such as the dissipation and mixing out of primary kinetic energy, and mixing out of static pressure gradients, play a larger role in the loss production.

The computational results gave significantly higher C''_{SKE} values than the experimental results. Taremi (2011) conducted a related high-speed study and also found that the computations overpredicted the C''_{SKE} to similar levels as those found here. While the computations predicted higher magnitudes of C''_{SKE} than those measured, both showed a comparable increase in C''_{SKE} due to the presence of the cavity. This suggested that while the CFD failed to accurately predict the loss production for such cavity flows, it did predict the increase in secondary kinetic energy production reasonably well. The increase in losses for the cavity case is partially attributed to the significant increase in the strength and size of the secondary vortical structures (due to the presence of the cavity and the development of the cavity vortex), particularly the passage and corner vortex which continue contributing to the loss production downstream. However, similar to the experiment results, the computational low values of C''_{SKE} relative to the computational $Y_{mixed-out}$ values, suggested that the dominant source of loss production is not the dissipation and mixing out of the secondary kinetic energy, but rather mixing out of primary kinetic energy, and static pressure gradients, play a larger role in the loss production.

5.7 Discussion and Conclusions

In conclusion, the results have shown that the presence of the upstream cavity noticeably altered the structure and the strength of the secondary flow. Oil surface flow visualization showed that the cavity results in complex flows that serve to strengthen the horse-shoe vortex giving rise to stronger passage and corner vortices downstream. The secondary kinetic energy increased significantly due to the presence of the cavity, and a major effect on the overall losses was also evident, with the cavity generating up to 14% higher losses ($Y_{mixed-out}$), when compared to the baseline flat endwall.

The computations mostly tended to overestimate the total losses, and secondary kinetic energy. They predicted almost equal loss levels in terms of $Y_{mixed-out}$ for both the flat and cavity endwalls. This suggests that while CFD can adequately predict the formation of the main vortical structures, it can lack in its ability to model the mixing processes leading to inaccurate total pressure loss estimations. However, while the secondary kinetic energy predictions were higher, the computations did predict an increase in the secondary kinetic energy due to the presence of the cavity, by a comparable factor to that observed in the experiment.

Previous low speed studies (e.g. Benner, 2003) have shown that secondary kinetic energy dissipation and mixing out is the dominant source of loss production, based on C''_{SKE} values being a substantial fraction of the $Y_{mixed-out}$ values. However, the results found here suggest that in high speed flows, the dissipation and mixing out of secondary kinetic energy is not the dominant source of loss production, and that other sources such as the dissipation and mixing out of primary kinetic energy, and mixing out of static pressure gradients, play a larger role in the loss production.

Although in the actual engine case the cavity will be ejecting purge flow, the conclusions reveal that just a simple upstream cavity can have a significant impact on the development of the secondary flows, with significant increases in the total pressure losses. From an engine design perspective, this suggests that the design of the platform overlap geometry may be strategically selected. The shape of the cavity can be optimized to attempt to control the recirculating flow within it, to minimize the formation of the “cavity” vortex, and control it to keep it from ejecting fluid into the blade passage and mixing with the pressure side leg of the horse-shoe vortex.

The cavity results in complex secondary flows, and purge flows would interact with these secondary flows possibly yielding an even more complex flow and perhaps even higher losses. The extent of purge flow effects on the current observed physics is not known.

However, the purge mass flow rate is small compared to the main gas path mass flow rate (e.g. 0.5%), and typically has relatively low momentum and may have a significant tangential component. Thus the purge flow may simply add to the fluid in the cavity vortex.

By understanding some of the basic flow physics provided here (e.g. formation of the cavity vortex, and strengthening of the pressure side leg of the horse-shoe vortex as a result) designers can seek ways to mitigate these losses by careful selection of platform overlap geometries or via endwall contouring to manipulate the local static pressure field. The next chapter presents the performance of an endwall contouring design, optimized for the cavity geometry presented, with the aim to reduce the total losses.

Endwall Contouring Performance at Design Mach Number

6.1 Introduction

Increased secondary losses are typically associated with increased blade loading. This has led to airfoil designs that can increase blade loading while maintaining relatively lower end-wall losses, named “high-lift” designs such as the present SL2P airfoil (see Section 3.5.1). To further reduce endwall losses, another recent development has been the use of endwall contouring: a combination of hills and valleys at the endwall, optimized to reduce the local cross-passage pressure gradient, thus reducing the secondary flows and losses.

In HP turbines, aspect ratios are considerably lower than LP turbines, and the effects of endwall losses are more significant. Thus, there is strong motivation to understand ways of controlling the endwall flows at the high speed conditions representative of HP turbines. In this chapter the performance of a non-axisymmetric endwall contouring design applied to an HP representative airfoil with an upstream cavity is assessed at design conditions. A discussion of its performance with regards to losses and flow physics is presented through both experimental and computational results.

6.2 Inlet Endwall Boundary Layer

Figure 6.1: Inlet endwall boundary layer comparison of flat, cavity, and contoured endwalls.

As was the case in Chapter 5, the upstream boundary layer was traversed at $2.0 C_x$ upstream of the leading edge of the cascade using the goose-neck pitot probe. Figure 6.1 shows the boundary layer profile in terms of the inlet Mach number ratio, $M_1/M_{1,ms}$. It was found that all three endwalls had very similar inlet endwall boundary layers. The integrated endwall boundary layer parameters, also presented in Figure 6.1, indicate essentially a zero pressure gradient turbulent boundary layer for all three cases.

6.3 Midspan Blade Loading Distributions

Figure 6.2 shows the midspan blade loading distributions for the contoured endwall, with the flat and cavity cases repeated from Chapter 5. As was the case with the flat and cavity endwalls, the results showed good agreement between the experiment and the PWA

Figure 6.2: Experimental and CFD (PWA design prediction) blade loading comparison for flat, cavity, and contoured endwalls.

design CFD prediction values, indicating a good match between the experiment and the design condition. Figure 6.2 shows that the midspan blade loading for the flat, cavity, and contoured endwalls are virtually identical. This was expected as all three endwalls share the same blade profile and suggests that any differences in the blade passage flow due to the cavity or contouring do not noticeably impact the midspan loading.

The author's own CFD post-diction blade loading results (shown in Figure 4.5) are virtually identical to the PWA CFD results and are thus not repeated here. However, detailed post-diction CFD results of the passage and downstream flow field are provided within Sections 6.4.3, and 6.5.4.

6.4 Inner-passage Flowfield Behaviour

6.4.1 Endwall Flow Visualization

Similarly to previous cases, oil surface flow visualization was conducted for the contoured endwall. The flow visualization method was explained in detail in Section 3.6. In Section 5.4.1 the baseline flat endwall was compared to the cavity endwall. In this section, the cavity endwall is compared to the contoured endwall.

As described in Section 2.5.6, the contouring makes use of streamline curvature to control the cross-passage pressure gradient in the blade passage in a favourable manner so as to reduce the endwall losses. Figure 6.3 shows a view comparing the cavity and contoured endwalls. The endwall contouring topology was presented in detail in Figure 3.11, but the main features are highlighted in the flow visualization figure. They are: a valley in middle of the passage entrance (V), a hill on the pressure side of the blade leading edge (H), a ridge in the middle of the passage (R - yellow dashed line), and a small trough near the pressure side of the blade towards the end of the passage (T - green dashed line).

Comparing the two endwalls, the strong separation lines associated with the escaped vortical fluid of the cavity vortex into the passage (previously discussed in Section 5.4.3) are highly visible on the cavity endwall, while appearing much weaker and less visible on the contoured endwall. This is highlighted by the flow region in the white dashed box for both endwalls on the figure. This can be attributed to the presence of the hill and the valley in that region. The hill serves to decrease the static pressure near the pressure side, while the valley increases the static pressure near the suction side resulting in reduction of the cross-passage pressure gradient near the endwall at the entrance of the passage. This reduction leads to reduced endwall cross flows, and hence less escaped vortical flow from the cavity vortex is mixed with the pressure side leg of the horse-shoe vortex and ultimately a weaker passage vortex is formed downstream. This same mechanism would also lead to a weaker formation of the pressure side leg of the horse-shoe vortex that takes place slightly above the separated region. Thus, the contouring can lead to a reduction in mixing the escaped recirculating flow from the cavity vortex, and the pressure side leg of the horse-shoe vortex.

Figure 6.3: Comparison of cavity, and contoured endwalls; endwall flow visualization with interpretation.

The hill also results in locally accelerating the flow in the hill region leading to the straightening of the limiting flow streamlines (relative to those of the cavity endwall) near the pressure side leading edge, as indicated by the white arrows near the leading edge at the entry to the passage. While the flow is eventually driven by the cross-passage pressure gradient from the pressure side to the suction side (as indicated by the shear stress trajectories), it is believed that this delay at the entry of the passage, serves to decrease the “feeding” of the lower momentum fluid to the downstream forming passage vortex, leading to a weaker vortex with lower secondary losses.

The endwall contouring appears also to have a noticeable impact near the outlet of the passage. Much stronger cross flows are apparent (indicated by the white arrows near the suction side at the trailing edge), than those on the cavity endwall. It is believed that these stronger cross flows will give rise to the development of a stronger corner vortex at the separation line marked by S_3 .

6.4.2 Suction Surface Flow Visualization

Figure 6.4 shows the suction surface flow visualization results. The cavity endwall (discussed previously in Chapter 5), shows two dominant separation lines, S_4 associated with the passage vortex, and S_{2S} associated with the suction side leg of the horse-shoe vortex.

The two dominant separation lines S_{2S} and S_4 are also visible on the contoured endwall. S_{2S} appears to migrate further along the passage than it did for the cavity endwall with a noticeably different shape. The S_4 separation line also appears to emerge later, at $0.45 \frac{S_s}{S_{s,max}}$, than it did for the cavity endwall, with a slightly smaller penetration depth. This is attributed to the weakening of the formation of the passage vortex due to a weaker mixing of the escaped cavity vortex vortical fluid with the pressure side leg of the horse-shoe vortex.

Figure 6.4: Comparison of cavity, and contoured endwalls; suction surface flow visualization with interpretation.

6.4.3 Computational Flow Physics and Visualization

Complementary CFD simulations at the design outlet Mach number of 0.80 were performed for all three endwalls. In this section the inner passage results for the cavity contoured endwall are examined to help in the understanding of the flow physics, while Section 6.5.4 presents the downstream results. The results for the flat and cavity endwalls were discussed in Chapter 5.

The computations revealed that features similar to those of the cavity endwall flow presented in Figure 5.9 were also largely present in the contoured endwall. However, the contouring did alter the Mach number distributions, as well as change the flow direction in the near endwall region. To illustrate this, Figure 6.5 shows the Mach number distributions overlaid with velocity vectors for a plane located 0.61 mm (1% span for the flat endwall) from the endwall surface, for both cases. The edge of the cavity, i.e. where the nominal endwall height begins to decrease, is marked with a dashed line, and the main features of the contouring are also overlaid for the contoured endwall (V: valley, H: hill, R: ridge, and T: trough).

The hill locally accelerated the flow as evidenced by the slightly higher Mach numbers approaching and downstream of the hill for the contoured endwall. This resulted in a

Figure 6.5: Computational results showing Mach number distributions with velocity vectors and contouring features, at 0.61 mm above the endwall surface

more axial flow near the endwall at the entrance of the passage in the region of the dashed square in the figure. This is consistent with the results observed in the experimental flow visualization that was presented in the previous sections. This “straightening” of the endwall flows at the passage inlet, as discussed earlier, can decrease the feeding of the lower momentum fluid into the downstream-forming passage vortex, leading to a weaker vortex with lower secondary losses.

The contouring also shows a noticeable acceleration on the suction side of the blade near the trailing edge, with reduced cross flows at the exit of the passage, and flow aligned more with the main streamwise flow direction. This is clear in the region highlighted by the dashed rectangle. Since the corner vortex is formed by such cross flows meeting the suction surface of the blade, the reduction in the cross flows might have led to a reduction in the corner vortex strength downstream. However, the experimental flow visualization results, suggest that the cross flows near the passage exit are in fact strengthened. The downstream results for the experiment and the computations are discussed in the next section.

6.5 Downstream Flow Field Behaviour

6.5.1 Loss Distributions

Seven-hole pressure probe measurements were made at $1.4 C_x$ from the leading edge (see Figure 3.3) for all three endwalls. Pressure losses, in the form of the local stagnation pressure loss coefficient (Y_{local}), and the pitchwise averaged stagnation pressure loss coefficient (Y_{pitch}) are presented here.

Figure 6.6: Comparison of baseline flat endwall, cavity endwall, and contoured endwall: (a) Downstream local stagnation pressure losses. (b) Pitchwise averaged stagnation pressure losses.

Figure 6.6 (a) shows Y_{local} contour flood results for all three endwalls. Comparison of the flat endwall to the cavity endwall was presented in Chapter 5, however all three endwalls are shown here for a complete comparison.

Looking at the contoured endwall, and comparing it to the cavity endwall, there is a similar loss pattern observed, but with a significant reduction in the loss region located at about 15% span (previously shown to be associated with the passage vortex), with the maximum loss core value reduced from 0.14 to 0.10. Also, the loss region near the endwall at about 2.5% span (previously shown to be associated with the corner vortex), increased, with a maximum loss core value increasing from 0.32 to 0.37. Thus, the endwall contouring appears to have noticeably reduced the losses associated with the passage vortex, but unfortunately in the course of weakening the passage vortex, the contouring seems to end up strengthening the cross flows near the trailing edge and thus increasing the strength and losses of the corner vortex. In the next section it will be shown that these changes in losses are indeed accompanied with changes in the downstream vortical structures.

Figure 6.6 (b) presents the pitchwise averaged pressure loss distributions. The most notable area of loss decrease occurs between 20% to 5% span which corresponds to the loss core that was previously shown to be associated with the passage vortex.

6.5.2 Streamwise Vorticity Field

Figure 6.7 (a) shows a colour flood of C_{ω_s} with contour lines of the local stagnation pressure loss coefficient (Y_{local}), for all three endwalls. All three endwalls are shown here for a complete comparison.

Comparing the contoured endwall to the other two cases, similar flow structures are observed. The three main vortical structures, the passage, counter, and corner vortices plus the near-endwall vorticity (also present in the cavity endwall) are all present in the contoured endwall. However, the counter vortex is significantly diminished in strength and size. Further, the passage vortex shape has changed (appears more spread out), and small increases in the strength of the corner vortex are observed.

Figure 6.7: Comparison of baseline flat endwall, cavity endwall, and contoured endwall: (a) Streamwise vorticity field with stagnation pressure loss contour lines. (b) Pitchwise averaged exit flow angle deviation.

The reduction in the size of the counter vortex is significant and somewhat unexpected, but a possible explanation for the reduction of the counter vortex is offered here. The contouring appears to reduce the amount of vortical flow that escapes from the cavity vortex into the passage (as indicated by the weaker separation lines in Figure 6.3). Therefore, there is reason to believe that the cavity vortex, as a result, might be larger in the contoured endwall than in the cavity endwall. A larger cavity vortex would likely trap more fluid from the suction side leg of the horse-shoe vortex through the mechanism previously illustrated in the enlarged area of Figure 5.9. As more of the fluid that eventually forms the counter vortex becomes part of the cavity vortex, the counter vortex size might be then reduced.

The change in the size and shape of the passage vortex is attributed to the change in escaped vortical flow from the cavity vortex and change in mixing between it and the pressure side leg of the horse-shoe vortex. A change that is a result of the modification of the cross-passage gradient caused by the contouring.

The small increase in the strength of the corner vortex may be attributed to the apparent strengthening of the cross flows towards the end of the passage (as observed in Figure 6.3). As was the case with the cavity endwall, the loss cores at 20%, 10%, and 2.5% span are in close proximity to the counter vortex, passage vortex, and corner vortex respectively. However, in the contoured endwall case, the centres of the vorticity for the passage and counter vortex were shifted slightly away from where the highest loss core values occurred. Further, there appears to be a disconnect between the vorticity strength and the drop in losses for the counter vortex and to a lesser degree the passage vortex. The counter vortex appears to have diminished significantly, with the losses near it remaining unchanged.

The counter vortex appears to have diminished significantly, with the losses near it remaining unchanged, but a possible explanation of this disconnect is suggested here. Vortices can generate losses by two mechanism: viscous shearing between each other, or between them and the primary fluid (i.e. fluid aligned with the streamwise direction). Thus, it is plausible that a diminished counter vortex might not translate into a drop in losses due to changes in the primary fluid surrounding it or the adjacent vortex (i.e. passage vortex). Further, a drop in losses might not necessarily be near the drop in vorticity, it could be that the diminished vorticity translates into a drop in losses in a region closer to a vortex with which it has interacted with (e.g. the passage vortex).

Figure 6.7 (b) shows the pitchwise averaged exit flow angle deviation (ψ) for all three endwalls. The contouring has only a small effect on the level of underturning (relative to the cavity endwall), causing a slight increase as can be seen from 20% to 10% span. Below

about 8% span, the contoured and the cavity endwall have very similar values of ψ , where the flow is mainly dominated by the near-endwall vorticity, which are very similar for both endwalls.

6.5.3 Secondary Kinetic Energy Field

To further understand the flow and the production of secondary losses, the secondary kinetic energy coefficient (C_{SKE}) is extracted from probe measurements made at $1.4 C_x$. Figure 6.8 (a) shows a contour plot of C_{SKE} , with secondary velocity vectors, and Y_{local} contour lines overlaid. The figure compares all three endwalls, with the same scale for direct comparison. The flat endwall and the cavity endwall were discussed in Chapter 5; but are repeated here for comparison purposes.

Comparing the contoured endwall to the cavity endwall, there are some similarities. The largest secondary velocities and C_{SKE} values occur in the vortex-vortex interaction regions, but at different parts of the interaction region. These regions are the region between the passage and counter vortex at about 15% span, and the region between the passage and counter vortex at about 3% span.

In the region about 15% span (between the counter and passage vortex), two regions are labelled as A and B for both the cavity and the contoured endwall. The contouring seems to have lowered the secondary kinetic energy significantly in region B, and has increased it in region A, consistent with the change in the passage vorticity distribution observed in the previous section. While the secondary kinetic energy has increased at A, overall the 15% span interaction region appears to have lower C_{SKE} for the contoured endwall.

The contoured endwall shows a significant reduction in C_{SKE} in the region between the passage and corner vortex (at about 3% span). This is attributed to the reduction in the size and strength of the counter vortex, and to a lesser extent the small changes in the passage vortex as was discussed in the previous section.

This decrease in secondary kinetic energy (specifically between 20% to 12% span) due to the contouring, is consistent with the small decrease in flow overturning as indicated by the flow angle deviation (ψ) previously presented, but repeated in Figure 6.8 (b). This decrease in flow turning due to the contouring, brings back the flow turning levels of the contouring endwall nearly to that of the baseline flat endwall between 20% to 12% span. Figure 6.9 (a), provides an alternative to Figure 6.8 (a), with the secondary velocity vectors replaced with secondary flow pseudo-streamlines. This further highlights the regions with the highest secondary kinetic energy.

Figure 6.8: Comparison of baseline flat endwall, cavity endwall, and contoured endwall: (a) Secondary kinetic energy field with stagnation pressure loss contour lines and secondary velocity vectors. (b) Pitchwise averaged exit flow angle deviation.

Figure 6.9 (b) shows the pitchwise mass-averaged C'_{SKE} for all three endwalls. As discussed previously, the presence of the cavity leads to a significant increase in secondary kinetic energy from about 20% span and below. The contoured endwall had lower sec-

ondary kinetic energy from about 20% to 10% span, where it reaches values comparable to the baseline flat case from 12% to 10% span. Below 10% both the cavity and the contoured endwalls have comparable values. Overall the endwall contouring resulted in a reduction of secondary kinetic energy with a planewise mass-averaged value (C''_{SKE}) of 0.0026, relative to 0.0058 for the cavity endwall, and 0.0017 for the baseline flat endwall.

Figure 6.9: Comparison of baseline flat endwall, cavity endwall, and contoured endwall: (a) Secondary kinetic energy field with stagnation pressure loss contour lines and secondary flow pseudo-streamlines. (b) Pitchwise averaged secondary kinetic energy.

As discussed previously, an important loss production mechanism, is the mixing out of the secondary flow structures and their associated secondary kinetic energy. Thus, the lower secondary kinetic energy of the contoured endwall suggests that the contoured endwall will generate lower losses downstream of $1.4 C_x$ than the cavity endwall, but likely still higher than the baseline flat endwall. This is further discussed in Section 6.6.

6.5.4 Computational Downstream Results

Complementary CFD simulations at the design outlet Mach number of 0.80 were performed for all three endwalls. Figure 6.10 shows a comparison of the losses in terms of the local stagnation pressure loss coefficient (Y_{local}) between the experimental results (a), and the computational results (b). The results for the contoured endwall are shown at the right of the figure.

Figure 6.10: Comparison of downstream local stagnation pressure losses: (a) Experimental results. (b) Computational results.

As was the case with the cavity endwall, the computations overall, showed significantly higher losses than the experiment. There were some broad similarities with the experiment, including a reduction in the loss core at about 10% span (coinciding with the passage vortex), with the computation showing the maximum loss core value decreasing from 0.23 to 0.15. A similar proportional reduction to that observed in the experiment, although the absolute values were higher than in the computations. The computations also showed

some increase in the loss area closest to the endwall, at 5% span and below (associated with the corner vortex), also in agreement with the trends observed in the experiment.

Figure 6.11 shows comparisons of the streamwise vorticity coefficient (C_{ω_s}), between the experimental results (a), and the computational results (b). As noted previously for the first two cases, the shape and size of the vortical structures were significantly different from those of the experiment. Nevertheless, the computational results yielded the same three main vortical structures; the passage vortex, counter vortex, and corner vortex, with loss cores typically associated with them. As was the case with the cavity computational results, the significantly enlarged near-endwall vorticity in the experiment was not predicted by the computations.

Figure 6.11: Comparison of streamwise vorticity field with stagnation pressure loss contour lines: (a) Experimental results. (b) Computational results.

The CFD results showed that the endwall contouring leads to a significant reduction in the strength of the passage vortex, and to a lesser extent the counter vortex, while

in the experiment most of the reduction was observed in the counter vortex, with some change in the shape of passage vortex. For the contoured endwall results, overall, larger and stronger passage, counter, and corner vortices were found in the computations than in the experiment (as was the case for the flat, and cavity endwalls).

As mentioned, one area of difference in the vorticity between the experimental and computational results was the near-endwall layer of vorticity. In the experiment this layer was clearly thickened by the presence of the cavity as seen in both the cavity and the contoured endwalls. This layer must be present in the baseline flat case but was likely too thin to have been measured, i.e. below the last traversing location at 2.5% span. The computations showed the thin layer of endwall vorticity, but failed to predict any thickening of it in the cavity and the contoured endwalls.

Generally, the near-endwall vorticity is produced by viscous effects, and once the vorticity has been generated it can be convected downstream where it can grow or shrink through mixing with other vortical fluid. In the computations, it was shown in Figure 5.9 that the escaped vortical fluid from the cavity vortex mixes with the pressure side leg of the horse-shoe vortex and lifts off the endwall early mid-passage (this applies to the contoured endwall, the contouring only reduces the amount of escaped vortical fluid). In the experiment, the separation lines from the flow visualization (see Figure 5.4) show that the escaped vortical fluid remains near the endwall until it meets the suction side surface of the blade (the contoured endwall also shows this, but with weaker separation lines). Therefore, in the experiment (for both the cavity and the contoured endwall) there might be more vortical fluid near the endwall than predicted by the computations. More vortical fluid near the endwall would add to the near-endwall vorticity through mixing leading to a thickening of the near-endwall vorticity layer. Thus, if the computations predicted less of a presence of vortical fluid near the endwall, it might explain the lack of thickening of the near-endwall vorticity layer.

Figure 6.12: Comparison of secondary kinetic energy field with stagnation pressure loss contour lines and secondary flow pseudo-streamlines: (a) Experimental results. (b) Computational results.

Figure 6.12 shows a comparison of C_{SKE} , with Y_{local} contour lines, and secondary flow pseudo-streamlines, between the experimental results (a), and the computational results (b). The computations showed much larger values of C_{SKE} than those observed in the experiment. The distribution of the secondary kinetic energy was significantly different than that of the experiment. However, there are some broad similarities. As was the case with the experiment, the computations showed the largest C_{SKE} values in the vortex-vortex interaction regions. Further, similar to the experiment, the effect of the contouring in the computations was a significant decrease of the secondary kinetic energy in the region between the passage vortex and the counter vortex. However, for the region between the passage vortex and the corner vortex, the secondary kinetic energy is confined to a smaller area but appears to have slightly increased. Overall, the endwall contouring did reduce the

secondary kinetic energy, as will be shown from the computed planewise mass-averaged secondary kinetic energy results presented in Section 6.6.

From Figure 6.11, it was clear that the presence of the cavity (in both the cavity endwall and the contoured endwall) did not effect the computed near-endwall vorticity (compared to the noticeable thickening observed in the experiment). However, the CFD as shown in Figure 6.12, did predict significant changes near the endwall in the secondary kinetic energy. Thus the computations seemed to capture the increase in secondary velocity as result of the cavity, but missed that this increased secondary velocity has vorticity associated with it.

Figure 6.13 compares the experimental and computational pitchwise averaged results for (a) losses (Y_{pitch}), (b) exit flow angle deviation (ψ), and (c) secondary kinetic energy coefficient (C'_{SKE}). The contoured endwall computations showed similar trends to the experiment until about 10% span. Below this, the computational results showed significantly higher losses than that of the experiment. The computational results for ψ showed similar trends to that of the experiment, where the contouring resulted in an increase in the level of underturning between 20% to 10% span. However, the computations predicted a much greater increase than that observed in the experiment, and overall the levels of underturning and overturning are higher in the computations then in the experiment.

The computations produced much higher values of secondary kinetic energy than the experiment did, as shown in Figure 6.13 (c). The contoured endwall computational results showed a similar trend to that of the experiment, with a large drop in the secondary kinetic energy between 20% to 10% span; a larger drop than observed in the experiment.

Figure 6.13: Comparison of pitchwise averaged results: (a) Pitchwise averaged stagnation pressure losses. (b) Pitchwise averaged exit flow angle deviation. (c) Pitchwise averaged secondary kinetic energy.

6.6 Total and Mixed-out Losses

The planewise mass-averaged stagnation pressure losses (Y_{total}), the mixed-out stagnation pressure losses ($Y_{mixed-out}$), and the planewise mass-averaged secondary kinetic energy coefficient (C_{SKE}'') are summarized in Figure 6.14. Both experimental and computational results are given.

Figure 6.14: Experimental and computational total losses, and secondary kinetic energy.

The experiment showed a 6.5% drop in losses (relative to the cavity endwall) as a result of the endwall contouring and a drop of about 10% when mixed-out. As was previously discussed in Chapter 5 the computations did not reflect the increase in losses due to the cavity as measured in the experiment. Similarly, for the contoured endwall, at 1.4 C_x the CFD yielded loss values comparable to that of the cavity endwall, and significantly higher than the cavity when mixed out. This further suggests that for complex flows involving such cavities and endwall contouring, RANS type CFD failed in its ability to model the mixing processes, leading to inaccurate total pressure loss predictions. Specifically, for this case, the CFD predicted opposite loss trends to those observed experimentally.

Experimental C''_{SKE} values of 0.0026 for the contoured endwall were found to be reduced by about 55% from the cavity endwall levels. The secondary kinetic energy is kinetic energy associated with the secondary velocity components. Thus, the lower experimental C''_{SKE} for the contoured endwall results is consistent with the reduction in secondary flows. The computational results predicted significantly higher magnitudes than the experimental

values, but with comparable reduction levels (about 49% lower than the cavity endwall levels). Taremi (2011) conducted a related high-speed study and also found that the computations overpredicted the C''_{SKE} to similar levels as those found here.

However, the large reduction in C''_{SKE} did not translate into a reduction of the mixed out losses, with the mixed-out losses relative to the cavity endwall actually increasing in the computations. This disconnect between secondary kinetic energy and mixed-out losses is of particular concern for endwall contouring. Some endwall contouring optimizations use C''_{SKE} as the objective function in selection of the final design, thus caution must be taken when selecting a design based on reduction of secondary kinetic energy. As was the case with the cavity endwall discussed in Chapter 5, this further suggests that in high speed flows, the dissipation and mixing out of secondary kinetic energy is not the dominant source of loss production.

6.7 Discussion and Conclusions

The experimental results have shown that the endwall contouring can successfully lower the losses in a high-speed linear cascade with an upstream cavity. Reduction levels based on mixed-out losses of about 10% relative to the cavity endwall case were observed. End-wall contouring (designed using the same Pratt & Whitney method used here) has been demonstrated to successfully reduce losses when applied to a *flat endwall* in a low-speed linear cascade, (Knezevici, 2011), with reduction levels of up to 10% achieved. A more recent study (Taremi, 2011) suggested that at high speeds, the endwall contouring applied to a flat endwall yielded a marginal reduction of losses relative to the flat endwall. Thus, the work in the present study suggests that the application of endwall contouring (at least at high-speeds) might be most beneficial in cases where secondary flows are unusually high (e.g. due to the presence of an upstream cavity).

This endwall contouring was designed by Pratt & Whitney using an optimization method described in Section 2.5.7. There are several ways of generating endwall contouring (as presented in Section 2.5.7), and the design arrived at by PWA is not necessarily the true optimum geometry. Thus, the reduction in losses measured in the experiment is not necessarily the best that could be achieved from endwall contouring. It is also worth noting that the author used the fully mixed-out losses to evaluate the benefits of the endwall contouring design, but in an actual engine, the flow would reach the next blade row long before it is fully mixed-out.

While this particular endwall contouring design was optimized using a RANS CFD solver for reduction of averaged total row losses, the CFD used a design level mesh (for quick convergence), a considerably coarser mesh than the one used in the computations presented here. The expected design performance based on the PWA optimization, was very marginal (Praisner (2005)). The computational results carried-out in this study used a significantly finer mesh and yielded no benefit for the endwall contouring. This suggests that in optimizing a contouring design, modelling considerations, such as mesh refinement, and turbulence models used, are of considerable importance in arriving at a suitable design.

From the computational study carried out here, two important conclusions pertinent to the design of endwall contouring can be stated. First, the apparent disconnect between the reduction in secondary kinetic energy and reduction of mixed-out losses, shows that caution must be used when using secondary kinetic energy as an objective function or a parameter in selection of the best contouring design. Second, the apparent disagreement between the experiment and the author's computations in assessing the endwall contouring performance (e.g. mixed-out losses), shows that solely relying on computational results to assess the performance of a design is not enough. In this study, the experiment showed a benefit, while the author's CFD did not. Further, as shown by Ingram *et al.* (2005), the opposite can also occur. In their study, the computations predicted a significant reduction

in losses, while the experiment showed an increase in losses due to an endwall separation not captured by the CFD.

The commercial CFD solver used in this study was ANSYS CFX 12; a mature and reliable solver that has been used by many researchers in the past. ANSYS CFX is a 2nd order accurate solver, thus truncation errors could account for some of the discrepancy between the computations and the experiment. However, the author believes that the disagreement in loss predictions is largely due to shortcomings in the turbulence modelling. As with all RANS-type CFD solvers, ANSYS CFX relies on a turbulence model. Turbulence is the underlying mechanism of energy exchange, and thus loss production in secondary flows. Therefore, any shortcomings in the turbulence model will result in inaccuracies in the loss predictions. The main shortcoming of the turbulence model used in this study (SST model), is that it is ultimately based on the Boussinesq approximation (as are most common turbulence models, so called “eddy-viscosity” models), which does not accurately describe the flow physics for complex flows as those seen in this work. An alternative to eddy-viscosity models, that do not use the Boussinesq approximation, are “Reynolds-stress models”. However, they are more computationally expensive, and are numerically unstable. Thus, they are not well suited for use in design-level computations.

Off-Design Mach Number Behaviour

7.1 Introduction

Gas turbine engines typically operate at their design condition for the majority of the time. However, during certain phases of operation (e.g. engine startup, aircraft climb or descent, etc.), the turbines may operate at off-design conditions, of both incidence and Mach number.

HP turbines typically see small variations in Mach number, and thus when investigating the effects of an upstream cavity geometry, and endwall contouring performance on the losses, it is pertinent to assess the off-design Mach number behaviour. In this chapter, experimental results at four off-design Mach numbers are discussed. The discussion includes both lower than design Mach number of 0.80 (0.75, and 0.69), and higher (0.84, and 0.89).

7.2 Results for Below Design Mach Number

7.2.1 Inlet Endwall Boundary Layer

	$M_{1,ms}$	$M_{2,is}$	$\delta_{99\%}$ (mm)	$\delta_{99\%}/h$	δ^* (mm)	θ (mm)	H
Flat	0.29	0.80	24.3	0.40	2.47	1.90	1.30
Cavity	0.28	0.80	25.5	0.42	2.44	1.90	1.29
Contoured	0.29	0.80	26.8	0.44	2.26	1.83	1.24
Flat	0.29	0.75	24.3	0.40	2.51	1.92	1.31
Cavity	0.28	0.75	25.5	0.42	2.54	1.97	1.29
Contoured	0.28	0.75	26.8	0.44	2.33	1.88	1.24
Flat	0.28	0.69	24.3	0.40	2.53	1.94	1.31
Cavity	0.27	0.69	25.5	0.42	2.60	2.01	1.29
Contoured	0.27	0.69	26.8	0.44	2.42	1.95	1.24

Figure 7.1: Inlet endwall boundary layer comparison of flat, cavity, and contoured endwalls, below design Mach number.

The upstream boundary layer was traversed at $2.0 C_x$ upstream of the leading edge of the cascade using a goose-neck pitot probe. This was done at design, and below design Mach numbers (0.80, 0.75, and 0.69) for all three endwalls. The inlet total pressure profile in the endwall boundary layer is required to determine the net loss production within the blade passage. Figure 7.1 shows the boundary layer profiles in terms of the inlet Mach number ratio, $M_1/M_{1,ms}$. It was found that all three endwalls have very similar inlet

endwall boundary layers at all three Mach numbers. The integrated endwall boundary layer parameters also presented in Figure 7.1 indicate a fully turbulent flow for all cases.

7.2.2 Midspan Blade Loading Distributions

Figure 7.2: Experimental and CFD (PWA design prediction) blade loading comparison for flat, cavity, and contoured endwalls below design Mach number.

Blade loading measurements were collected at design, and below design Mach numbers (0.80, 0.75, and 0.69). Figure 7.2 shows that below the design Mach number, the outlet Mach number has significant impact on the suction surface, and a small but noticeable effect on the pressure surface towards the trailing edge, particularly at the lowest Mach number of 0.69.

7.2.3 Downstream Loss Distributions

Seven-hole pressure probe measurements were made at $1.4 C_x$ from the leading edge (see Figure 3.3) for all three endwalls, at design, and below design Mach numbers (0.80, 0.75, and 0.69). Figure 7.3 shows Y_{local} contour flood results for all cases. As usual, the view is of the measurement plane viewed looking upstream with the passage pressure side on the left, and the passage suction side on the right.

Figure 7.3: Downstream local stagnation pressure losses, below design Mach number: (a) Flat endwall. (b) Cavity endwall. (c) Contoured endwall.

Overall the loss patterns are very similar, with an apparent slight increase in the overall loss (in the secondary flow region) for all three endwalls as the Mach number decreases. For the flat endwall, a slight increase was noticed in the main loss core at 20% span, with the maximum loss value increasing from 0.35 to 0.36. In the smaller loss core near the endwall at 2.5% span, a slight drop in the maximum loss core value was noticed, from 0.26 to 0.24. For the cavity endwall, higher losses were seen in the loss region between 8% to

15% span, with a maximum loss value increasing from 0.14 to 0.16, while maximum loss core values at 20% and 2.5% span appeared unchanged. For the contoured endwall, the maximum loss value at the loss core near 20% span slightly increased from 0.32 to 0.33, and increased from 0.10 to 0.13 for the loss region between 8% to 15% span, while the loss region near the endwall at 2.5% span appeared smaller, with a maximum loss value dropping from 0.37 to 0.33.

The loss-contour results suggest that the decrease in Mach number led to an increase in losses in the main loss region between about 22% to 8% span. This was more apparent for the cavity, and contoured endwall cases. A mechanism for the generation of secondary losses is the viscous shearing between oppositely rotating vortices. It may be that the diffusion of the secondary flows associated with the oppositely rotating passage and counter vortices leads to stronger viscous shearing between the them, leading to the increase in losses in that region.

The loss region near the endwall at 2.5% span (previously shown to be associated with the corner vortex), slightly decreased. This is attributed to a reduction in size and strength of the corner vortex as presented in the next section.

7.2.4 Downstream Streamwise Vorticity Field

Streamwise vorticity (C_{ω_s}) can be used to identify the vortical structures and quantify their strength and sense of rotation. C_{ω_s} was extracted from seven-hole pressure probe measurements that were made at $1.4 C_x$ from the leading edge.

Figure 7.4 shows a colour flood of C_{ω_s} with contour lines of the local stagnation pressure loss coefficient (Y_{local}), for: (a) the flat endwall, (b) the cavity endwall, and (c) the contoured endwall. The three main vortical structures were found for all three endwalls: the counter clockwise rotating passage vortex, the clockwise rotating counter vortex, and the clockwise rotating corner vortex.

Figure 7.4: Streamwise vorticity field with stagnation pressure loss contour lines, below design Mach number: (a) Flat endwall. (b) Cavity endwall. (c) Contoured endwall.

The most discernable trend common to all three endwalls, is the apparent reduction in strength and size of the corner vortex as the Mach number decreases, which is perhaps accompanied by a local decrease in loss as seems to be shown in Figure 7.3. For the flat endwall, and cavity endwall, very little change in the passage and counter vortices was noticeable. In the contoured endwall however, the counter vortex appeared to be diminishing slightly with the decrease in Mach number, and the passage vortex seems

strongest at Mach 0.75. The near-endwall vorticity present in the cavity and the contoured endwall cases, appears to reduce in size as the Mach number decreases, with the contoured endwall showing the most noticeable reduction. Overall, the vortical structures of the contoured endwall appeared to be the most sensitive to the decreasing Mach number.

7.2.5 Downstream Secondary Kinetic Energy Field

The secondary kinetic energy coefficient (C_{SKE}) is also extracted from probe measurements. Figure 7.5 shows a contour plot of C_{SKE} , with Y_{local} contour lines overlaid.

The results show that the secondary kinetic energy increases as the Mach number is decreased. The distribution pattern remained the same, with the highest values concentrated in the vortex-vortex interaction regions. These are regions between the oppositely rotating passage and counter vortex at about 15% span, and in between the oppositely rotating passage and corner vortex (and near-endwall vorticity also, for the cavity endwall and contoured endwall) at about 3% span.

This increase in secondary kinetic energy is attributed to the vortical structures diffusing more, due to lower momentum in the primary flow direction at the lower Mach numbers, thus allowing for larger secondary kinetic energy between the vortical structures.

Figure 7.5: Secondary kinetic energy field with stagnation pressure loss contour lines, below design Mach number: (a) Flat endwall. (b) Cavity endwall. (c) Contoured endwall.

7.3 Results for Above Design Mach Number

7.3.1 Inlet Endwall Boundary Layer

Figure 7.6 shows the boundary layer profile in terms of the inlet Mach number ratio, $M_1/M_{1,ms}$. As was the case for the lower Mach numbers, it was found that all three endwalls have very similar inlet endwall boundary layers. The integrated endwall boundary

	$M_{1,ms}$	$M_{2,is}$	$\delta_{99\%}$ (mm)	$\delta_{99\%}/h$	δ^* (mm)	θ (mm)	H
Flat	0.29	0.80	24.3	0.40	2.47	1.90	1.30
Cavity	0.28	0.80	25.5	0.42	2.44	1.90	1.29
Contoured	0.29	0.80	26.8	0.44	2.26	1.83	1.24
Flat	0.29	0.84	25.5	0.42	2.58	1.99	1.30
Cavity	0.29	0.84	25.5	0.42	2.58	2.01	1.28
Contoured	0.29	0.84	26.8	0.44	2.50	2.02	1.23
Flat	0.30	0.89	25.5	0.42	2.54	1.95	1.30
Cavity	0.29	0.89	26.8	0.44	2.52	1.96	1.28
Contoured	0.30	0.89	25.5	0.42	2.39	1.94	1.24

Figure 7.6: Inlet endwall boundary layer comparison of flat, cavity, and contoured endwalls, above design Mach number.

layer parameters shown in Figure 7.6, indicate fully-turbulent flow, and essentially a zero-pressure gradient boundary layer for all cases.

7.3.2 Midspan Blade Loading Distributions

Above the design Mach number, it is evident from Figure 7.7, that the outlet Mach number has significant impact on the suction surface, and a small but noticeable effect on the pressure surface towards the trailing edge, particularly at the highest Mach number of 0.89.

Figure 7.7: Experimental and CFD (PWA design prediction) blade loading comparison for flat, cavity, and contoured endwalls above design Mach number.

The increase in Mach number seems to increase the blade loading at mid to aft of the passage, but appears to be less effected near the end of the passage (note some measurement points are missing due to unintentional blocking of the static taps during testing).

7.3.3 Downstream Loss Distributions

Figure 7.8 shows Y_{local} contour flood results for: (a) the flat endwall, (b) the cavity endwall, and (c) the contoured endwall, in order of increasing Mach number from left to right.

Overall, the patterns remained very similar, with an apparent decrease in the overall loss (in the secondary-flow region) for all three endwalls as the Mach number increases. For the flat endwall, a decrease was noticed in the main loss core at 20% span, with the maximum loss value dropping from 0.35 to 0.31. In the smaller loss core near the endwall at 2.5% span, a drop in the maximum loss core value was noticed, from 0.26 to 0.21.

For the cavity endwall, lower losses were seen in the loss region between 8% to 15% span, with a maximum loss value decreasing from 0.14 to 0.12, while the maximum loss core value at 20% span dropped from 0.32 to 0.30. The loss core region at 2.5% span appeared to be mostly unchanged. For the contoured endwall, the maximum loss value at

Figure 7.8: Downstream local stagnation pressure losses, above design Mach number: (a) Flat endwall. (b) Cavity endwall. (c) Contoured endwall.

the loss core near 20% span slightly decreased from 0.32 to 0.31, and decreased from 0.10 to 0.09 for the loss region between 8% to 15% span. The loss region near the endwall at 2.5% span decreased with a maximum loss value dropping from 0.37 to 0.33.

The loss contours suggest that the increase in Mach number results in a decrease in losses in the secondary flow region below 30% span, with the most noticeable effect apparent for the contoured endwall, particularly in the region between 8% to 15% span

at 40% pitch. This is attributed to the “concentration” of secondary flows as the primary momentum increases with Mach number.

As previously mentioned, the generation of secondary losses was partially due to the viscous shearing between oppositely rotating vortices. It may be that the stretching of a vortex as it is convected along an accelerating flow with increasing Mach number, has the effect of increasing its strength near its core (and perhaps weakening it at its outer regions) with little change in the overall shape, a sort of “concentration” of the vorticity. Thus, leading to weaker viscous shearing between the them, resulting in a decrease in losses in that region. These vortices are further discussed in the next section.

7.3.4 Downstream Streamwise Vorticity Field

Figure 7.9 shows a colour flood of C_{ω_s} with contour lines of the local stagnation pressure loss coefficient (Y_{local}), for: (a) the flat endwall, (b) the cavity endwall, and (c) the contoured endwall. All contour plots have the same scale for direct comparison, with values beyond the range of the scale shown by the red arrows.

For the flat endwall, all three vortical structures appeared to be concentrated more as indicated by their local increased strength and small change in shape, consistent with the increase in primary flow momentum with increasing Mach number. For the cavity endwall, and the contoured endwall, the most discernable trend was the apparent large increase in strength (and to a lesser extent apparent size) of the corner vortex as the Mach number increases. The counter vortex seemed to be commonly effected by the increasing Mach number for all three endwalls.

The shape of the near-endwall vorticity changed with the increasing Mach number. This vorticity layer spanning the entire pitch at the design Mach number significantly changes in shape. This is most notable in the cavity endwall case shown in Figure 7.9 (b).

Figure 7.9: Streamwise vorticity field with stagnation pressure loss contour lines, above design Mach number: (a) Flat endwall. (b) Cavity endwall. (c) Contoured endwall.

This further suggests that formation of the cavity vortex can have a significant impact on the downstream secondary flows with changes in Mach number.

7.3.5 Downstream Secondary Kinetic Energy Field

Figure 7.10 shows a contour plot of C_{SKE} , with Y_{local} contour lines overlaid for: (a) the flat endwall, (b) the cavity endwall, and (c) the contoured endwall.

Figure 7.10: Secondary kinetic energy field with stagnation pressure loss contour lines, above design Mach number: (a) Flat endwall. (b) Cavity endwall. (c) Contoured endwall.

The results show that the secondary kinetic energy decreases as the Mach number is increased. The distribution pattern remained the same, with the highest values concentrated in the vortex-vortex interaction regions.

As the Mach number increases, the primary momentum increases, and thus the vortical structures are stretched as they are convected along an accelerating flow. This can lead to an increase in the vorticity strength near its core, and perhaps strengthening its tangential

velocity at its outer regions. The slight drop in C_{SKE} might be attributed to the changes in the tangential velocities of the oppositely rotating counter, and passage vortices.

7.4 Total Row Losses, and Loss Breakdown

Figure 7.11: Total losses and loss break-down; decreasing mach number (black to blue), and increasing Mach number (black to red).

The planewise averaged (over the 1.4 C_x measurement plane) stagnation pressure losses (Y_{total}), and the mixed-out stagnation pressure losses ($Y_{mixed-out}$) are summarized in Figure 7.11. Experimental results, with the standard decomposition of the losses into profile and secondary losses (as detailed in Section 3.7) for all three endwalls are given.

Figure 7.12: Variation of mixed-out losses with Mach number for all three endwall cases.

The majority of the change in losses for both above and below the design Mach number was in the secondary losses as indicated by the bar graphs in Figure 7.11, with the cooler colours for below design Mach number (0.75, and 0.69), and warmer colours for above design Mach number (0.84, and 0.89).

Figure 7.12 shows the variation of the mixed-out losses with Mach number for all three endwall case, and indicates that the losses decrease with increasing Mach number. The most notable change occurs slightly above and slightly below the design Mach number, with the cavity endwall showing the most drastic decrease as the Mach number is increased from 0.80 to 0.84. The results also suggest that the drop in losses plateaus as the Mach number increases further.

The endwall contouring continued to reduce losses (relative to the cavity endwall) below and above the design Mach number. However, as shown in Figure 7.12, the endwall contouring performance (i.e. drop in losses relative to the cavity endwall levels) appears to be best at about Mach 0.80.

7.5 Discussion and Conclusions

It was found that the changes in Mach number can impact the flow for all three endwalls. Overall, losses increased as the Mach number decreased, and decreased as the Mach number increased. The increased losses were mostly accompanied by increased secondary kinetic energy (below design Mach number), while the decreased losses were accompanied by mostly decreased secondary kinetic energy (above design Mach number).

The experimental results shown here can offer some insights into the “robustness” of non-axisymmetric endwall contouring as a viable technology for use in high-pressure (HP) turbines. I.e. does it only work well when operating at the design point that the contouring was designed for, or is the design robust enough to continue performing at off-design conditions.

Off-design conditions, can take three main forms: (i) off-design incidence, (ii) off-design Mach number, and (iii) off-design Reynolds number. In this work, the incidence was not varied. Further, as the outlet Mach number was varied by controlling the upstream blowing pressure, the Reynolds number is also varied along with the Mach number. Corriveau (2005) and Hall (2012) carried out profile loss studies using mostly the same experimental setup as the work presented here, and have shown results that suggest that where changes in profile losses were observed, the change in Reynolds number was a significant contributor. The profile losses remained almost constant for the range of Mach numbers investigated in this study, thus there is reason to believe that the loss trends observed in this study while varying the Mach number are primarily a function of the change in Mach number.

The results revealed that, non-axisymmetric endwall contouring can be successfully used to reduce losses at small off-design Mach numbers. However, this particular design (a PWA design), based on the mixed-out loss values, appeared to perform best at or near

the design Mach number. It was capable of reducing losses by about 10% (relative to the cavity endwall) at Mach 0.80, with the performance deteriorating to about 8% at Mach 0.69, and 7% at Mach 0.89.

Conclusions, Contributions, and Recommendations

8.1 Conclusions

The research undertaken aimed to contribute to the understanding of secondary flows in HP turbine blade passages. More specifically, to improve the understanding of vortical structures near the endwall in the presence of an upstream cavity, in the context of modern airfoil designs at transonic speeds. Further, it aimed to assess the viability of using non-axisymmetric endwall contouring to reduce endwall losses including those generated by the presence of an upstream cavity. The endwall contouring study included the use of a modern airfoil, at HP turbine representative flow speeds, and addressed to some degree the “robustness” of the design by examining off-design Mach number performance.

To achieve these aims, transonic linear cascade testing was used with one blade geometry (SL2P), and three different endwalls: a flat baseline endwall, an endwall with an upstream cavity, and a contoured endwall with the same upstream cavity. Seven-hole pressure probe downstream measurements were collected at Mach 0.69, 0.75, 0.80 (design), 0.84, and 0.89. Oil-dye surface flow visualization was done on the endwall and blade sur-

faces at the design Mach number of 0.80. Complementary CFD was also carried out using the commercial CFD software ANSYS CFX 12 for all three endwalls at the design Mach number.

The results have shown that the presence of an upstream cavity can noticeably alter the structure and the strength of the secondary flow. Oil surface flow visualization and computational visualization showed that the cavity results in the formation of a “cavity vortex” that serves to strengthen the pressure side leg of the horse-shoe vortex, giving rise to a larger and stronger passage vortex downstream. When compared to the baseline flat endwall, measurements downstream showed a substantial increase in the size and strength of the passage vortex, while the counter vortex and the corner vortex were also strengthened, but to a lesser degree. The presence of the cavity also introduced a significant increase in the level of overturning due to the reduction of the near endwall momentum caused by the presence of the cavity. The secondary kinetic energy increased significantly due to the presence of the cavity. A major effect on the overall losses was also evident, with the cavity endwall generating up to 14% higher losses (mixed-out row losses), relative to the baseline flat endwall.

Further, the results have shown that the endwall contouring successfully lowers the losses by altering the structure and the strength of the secondary flows. Surface flow visualization showed that the contouring reduces the amount of escaped vortical fluid from the cavity vortex entering the passage. The endwall contouring resulted in a complex flow, but with the end results being the reduction of secondary kinetic energy, and a total reduction in losses relative to the cavity endwall. Compared to the cavity endwall, the contoured endwall results showed a significant reduction in the size and strength of the counter vortex, accompanied by a large reduction in the secondary kinetic energy in the vortex-vortex regions. Overall, the contoured endwall showed a 6.5% drop in total row losses as a result of the endwall contouring and a drop of about 10% mixed-out.

The computations mostly tended to overestimate the total losses and secondary kinetic energy. They predicted almost equal loss levels in terms of $Y_{mixed-out}$ for both the flat and cavity endwalls, and failed to predict any benefit for the endwall contouring.

Both the experimental and computational results suggest that in high speed flows, the dissipation and mixing out of secondary kinetic energy is not the dominant source of loss production, and that other sources such as the dissipation and mixing out of primary kinetic energy, and mixing out of static pressure gradients, play a larger role in the loss production. Thus, caution must be applied when optimizing endwall contouring designs using secondary kinetic energy as the objective function for complex cavity flows such as the ones in this study. Further, the apparent disagreement between the experiment and the author's computations in assessing the endwall contouring performance (e.g. mixed-out losses), shows that solely relying on computational results to assess the performance of a design is not sufficient.

In assessing the off-design performance of the endwall contouring, the experimental results have shown that the endwall contouring can continue to successfully reduce the losses at small off-design Mach numbers. It was found that the changes in Mach number can impact the flow for all three endwalls, but the cavity and contoured endwall appeared more sensitive to the changes in Mach number. HP turbines typically see relatively small variations in Mach number, and the results here suggested that endwall contouring designed for an upstream cavity, is a viable option for reducing losses even at slightly off-design Mach numbers.

8.2 Contributions

One of the main contributions made in this work is the understanding of some of the basic flow physics associated with the cavity flows and the mechanism behind the loss

generation (e.g. formation of the cavity vortex, and strengthening of the pressure side leg of the horse-shoe vortex and the passage vortex downstream as a result).

Secondly, the work here has experimentally validated a new design method (Pratt & Whitney method) for generating non-axisymmetric endwall contouring. Specifically, the application of the non-axisymmetric endwall contouring design was proved to successfully reduce the secondary losses in an endwall with an upstream cavity using a modern “high-lift” blade profile (SL2P). The robustness of the design was also partly assessed by examining the contouring performance at off-design Mach numbers.

Some contributions towards arriving at a better metric for optimizing endwall contouring were made. While a new metric was not proposed, it was shown that secondary kinetic energy is not a good choice for optimizing endwall contouring in high-speed flows as used by some contouring design methods.

Through these contributions, designers can seek ways to mitigate the cavity losses by careful selection of platform overlap geometries and combining them with endwall contouring to further lower losses. Recommendations towards this goal are outlined next.

8.3 Recommendations

In actual engine high-pressure (HP) turbine configurations, the cavity at the rotor-stator interface (see Figure 1.2) has cold flow bled from the compressor, known as purge flow, for use in cooling the rotor disk, and preventing ingestion of the hot combustion gases into the disk space. The results from the present work have shown that the mere presence of the cavity results in complex secondary flows, and purge flows would likely interact with these secondary flows yielding an even more complex flow with perhaps even higher losses. Thus, it is recommended that this work be extended to include the effect of purge flow in terms of understanding the flow physics as well as determining its impact on losses.

With the understanding of the effects of purge flow, the shape of the cavity can be optimized to attempt to control the recirculating flow within it, to minimize the formation of the cavity vortex, and perhaps to limit it from entering the blade passage and mixing with the pressure side leg of the horse-shoe vortex. Naturally, non-axisymmetric endwall contouring would also need to be revisited with the effects of purge flow taken into consideration in terms of newly optimized designs and their performance in reducing losses. Further, designs that extend the endwall contouring upstream beyond the blade leading edge and into the cavity can also be evaluated. It is worth noting that some research at low speeds that takes into account the effect of purge has been done, including on-going research at Carleton university (see MacIsaac (2011)), but very little research has been done at higher Mach numbers representative of those in HP turbines.

In optimizing such designs, further studies must be undertaken to arrive at an objective function that better reflects the actual performance of the design. From the comparison between the experimental and computational results in this study, it was evident that the CFD results do not accurately capture the diffusion and mixing of the secondary flow structures accurately, resulting in inaccurate pressure loss predictions. Thus, additional numerical studies are recommended, utilizing experimental results for validation, with the aim to improve the CFD turbulence models to achieve more accurate loss predictions.

Lastly, the HP turbine typically sees small variations of off-design incidence, and their effects on the cavity flow, and endwall contouring should be assessed. Thus, it is recommended that the effect of engine representative variations in inlet incidence be investigated. Such investigations could yield better understanding of the flow physics, leading to a more robust and optimized platform overlap geometries and endwall contouring designs.

List of References

- ABRAHAM, S., PANCHAL, K., EKKAD, S.V., NG, W., BROWN, B.J., & MALANDRA, A. 2011. Effect of Turbine Airfoil Shape on Aerodynamic Losses for Turbine Airfoils Operating Under Transonic Conditions. *Proceedings of the ASME Turbo Expo, Paper No. GT2011-45188*.
- ANKER, J.E., & MAYER, J.F. 2002. Simulation of the Interaction of Labyrinth Seal Leakage Flow and Main Flow in an Axial Turbine. *Proceedings of the ASME Turbo Expo, Paper No. GT-2002-30348*.
- ATKINS, M.J. 1987. Secondary Losses and End-Wall Profiling in a Turbine Cascade. *IMechE Paper No. C255/87*.
- AUNAPU, N.V., VOLINO, R.J., & STODDARD, R.M. 2000. Secondary Flow Measurements in a Turbine Passage with Endwall Flow Modification. *ASME Journal of Turbomachinery*, **122**, 651–658.
- BAGSHAW, D.A., INGRAM, G.L., GREGORY-SMITH, D.G., & STOKES, M.R. 2005. An Experimental Study of Reverse Compound Lean in a Linear Turbine Cascade. *IMechE Journal of Power and Energy*, **219**, 443–449.

- BAGSHAW, D.A., INGRAM, G.L, GREGORY-SMITH, D.G., STOKES, M.R., & HARVEY, N.W. 2008a. The Design of Three-Dimensional Turbine Blades Combined with Profiled Endwalls. *IMechE Journal of Power and Energy*, **222**, 93–102.
- BAGSHAW, D.A., INGRAM, G.L, GREGORY-SMITH, D.G., STOKES, M.R., & HARVEY, N.W. 2008b. An Experimental Study of Three-Dimensional Turbine Blades Combined with Profiled Endwalls. *IMechE Journal of Power and Energy*, **222**, 103–110.
- BASSI, F., OSNAGHI, C., PERDICHIZZI, A., & SAVINI, M. 1989. Secondary Flows in a Transonic Cascade: Comparison Between Experiments and Numerical Results. *ASME Journal of Fluids Engineering*, **111**, 369–377.
- BEAUDOIN, J.F, CADOT, O., AIDER, J.-L., & WESFREID, J.E. 2004. Three-Dimensional Stationary Flow over a Backward-Facing Step. *European Journal of Mechanics B/Fluids*, **23**, 147–155.
- BECZ, S., MAJEWSKI, M.S., & LANGSTON, L.S. 2004. An Experimental Investigation of Contoured Leading Edges for Secondary Flow Loss Reduction. *Proceedings of the ASME Turbo Expo, Paper No. GT2004-53964*.
- BENNER, M.W. 2003. *The Influence of Leading-Edge Geometry on Profile and Secondary Losses in Turbine Cascades*. Doctoral Thesis, Carleton University, Ottawa, Canada.
- BENNER, M.W., SJOLANDER, S.A., & MOUSTAPHA, S.H. 2006a. An Empirical Prediction Method for Secondary Losses in Turbines: Part I - A New Loss Breakdown Scheme and Penetration Depth Correlation. *ASME Journal of Turbomachinery*, **128**, 273–280.
- BENNER, M.W., SJOLANDER, S.A., & MOUSTAPHA, S.H. 2006b. An Empirical Prediction Method for Secondary Losses in Turbines: Part II - A New Secondary Loss Correlation. *ASME Journal of Turbomachinery*, **128**, 281–291.

- BINDER, A., & ROMNEY, R. 1983. Secondary Flow Effects and Mixing of the Wake Behind a Turbine Stator. *ASME Journal of Engineering for Power*, **105**, 40–46.
- BOLETIS, E. 1985. Effects of Tip Endwall Contouring on the Three-Dimensional Flow Field in an Annular Turbine Nozzle Guide Vane: Part 1—Experimental Investigation. *ASME Journal of Engineering for Gas Turbines and Power*, **107**, 1033–1040.
- BOYLE, R.J., & HAAS, J.E. 1982. Comparison of Experimental and Analytic Performance for Contoured Endwall Stators. *Proceedings of the 18th Joint Propulsion Conference, Paper No. NASA-TM-82877*.
- BRENNAN, G., HARVEY, N.W., ROSE, M.G., FOMISON, N., & TAYLOR, M.D. 2001. Improving the Efficiency of the Trent 500 HP Turbine Using Non-Axisymmetric End Walls: Part 1 Turbine Design. *Proceedings of the ASME Turbo Expo, Paper No. 2001-GT-0444*.
- BROWN, L.E. 1972. Axial Flow Compressor and Turbine Loss Coefficients: A Comparison of Several Parameters. *Journal of Engineering for Power*, **94A**, 193–201.
- BURD, S.W., & SIMON, T.W. 2000. Effects of Slot Bleed Injection over a Contoured Endwall on Nozzle Guide Vane Cooling Performance: Part I - Flow Field Measurements. *Proceedings of the ASME Turbo Expo, Paper No. 2000-GT-199*.
- CAMPBELL, D.A. 1978. Gas Turbine Disc Sealing System Design. *Proceedings of the AGARD, Paper No. AGARD-CP-237 18*.
- CAMUS, J.-J., DENTON, J.D., SOULIS, J.V., & SCRIVENER, C.T.J. 1984. An Experimental and Computational Study of Transonic Three-Dimensional Flow in a Turbine Cascade. *ASME Journal of Engineering for Gas Turbines and Power*, **106**, 414–420.
- CANON. 2002 (September). *PowerShot G3 Digital Camera User Guide*.

- CAO, C., CHEW, J.W., MILLINGTON, P.R., & HOGG, S.I. 2003. Interaction of Rim Seal and Annulus Flows in Axial Flow Turbine. *Proceedings of the ASME Turbo Expo*, Paper No. GT2003-38368.
- CHERRY, D., WADIA, A., BEACOCK, R., SUBRAMANIAN, M., & VITT, P. 2005. Analytical Investigation of a Low Pressure Turbine with and Without Flowpath Endwall Gaps, Seals and Clearance Features. *Proceedings of the ASME Turbo Expo*, Paper No. GT2005-68492.
- CHUNG, J.T., & SIMON, T.W. 1993. Effectiveness of the Gas Turbine Endwall Fences in Secondary Flow Control at Elevated Freestream Turbulence Levels. *ASME Paper No. 93-GT-51*.
- CHUNG, J.T., SIMON, T.W., & BUDDHAVARAPU, J. 1991. Three-Dimensional Flow Near the Blade/Endwall Junction of Gas Turbine: Application of a Boundary Layer Fence. *ASME Paper No. 91-GT-45*.
- CORRIVEAU, D. 2005. *Influence of Loading Distribution on the Performance of High Pressure Turbine Blades*. Doctoral Thesis, Carleton University, Ottawa, Canada.
- CORRIVEAU, D., & SJOLANDER, S.A. 2002. Impact of Flow Quality in Transonic Cascade Wind Tunnels: Measurements in an HP Turbine Cascade. *Proceedings of the 23rd International Congress of Aeronautical Science*.
- CUMPSTY, N.A., & HORLOCK, J.H. 2006. Averaging Nonuniform Flow for a Purpose. *ASME Journal of Turbomachinery*, **128**, 120–129.
- CURTIS, E.M., HODSON, H.P., BANIEGHBAL, M.R., DENTON, J.D., HOWELL, R.J., & HARVEY, N.W. 1997. Development of Blade Profiles for Low-pressure Turbine Applications. *ASME Journal of Turbomachinery*, **119**, 531–538.

- DANIELS, W.A., JOHNSON, B.V., GRABER, D.J., & MARTIN, R.J. 1992. Rim Seal Experiments and Analysis for Turbine Applications. *ASME Journal of Turbomachinery*, **114**, 426–432.
- DE LA ROSA BLANCO, E., HODSON, H.P., & VAZQUES, R. 2005. Effect of Upstream Platform Geometry on the Endwall Flows of a Turbine Cascade. *Proceedings of the ASME Turbo Expo, Paper No. GT2005-68938*.
- DEICH, M.E., ZARYANKIN, A.E., FILLIPOV, G.A., & ZATSEPIN, M.F. 1960. Method of Increasing the Efficiency of Turbine Stages with Short Blades. *Teploenergetika*, **7**, 18–24.
- DENTON, J.D. 1993. Loss Mechanisms in Turbomachines. *ASME Journal of Turbomachinery*, **115**, 620–650.
- DIPPOLITO, G., DOSSENA, V., & MORA, A. 2011. The Influence of Blade Lean on Straight and Annular Turbine Cascade Flow Field. *ASME Journal of Turbomachinery*, **133**, 011013 1–9.
- DORFNER, C., NICKE, E., & VOSS, C. 2007. Axis-Asymmetric Profiled Endwall Design Using Multiobjective Optimization Linked with 3D RANS-Flow-Simulations. *Proceedings of the ASME Turbo Expo, Paper No. GT2007-27268*.
- DOSSENA, V., PERDICHIZZI, A., & SAVINI, M. 1999. The Influence of Endwall Contouring on the Performance of a Turbine Nozzle Guide Vane. *ASME Journal of Turbomachinery*, **121**, 200–208.
- DOSSENA, V., D’IPPOLITO, G., & PESATORI, E. 2004. Stagger Angle and Pitch-Chord Ratio Effects on Secondary Flows Downstream of a Turbine Cascade at Several Off-Design Conditions. *Proceedings of the ASME Turbo Expo, Paper No. GT2004-54083*.

- DRUCK. 1994. *Druck DPI 605 User Manual K136*.
- DUDEN, A., RAAB, I., & FOTTNER, L. 1999. Controlling the Secondary Flow in a Turbine Cascade by Three-Dimensional Airfoil Design and Endwall Contouring. *ASME Journal of Turbomachinery*, **121**, 191–199.
- EYMAN, S., REINMÖLLER, U., NIEHUIS, R., FÖRSTER, W., BEVERSDORFF, M., & GIER, J. 2002. Improving 3D Flow Characteristics in a Multistage LP Turbine by Means of Endwall Contouring and Airfoil Design Modification – Part 1: Design and Experimental Investigation. *Proceedings of the ASME Turbo Expo, Paper No. GT-2002-30352*.
- GENTILHOMME, O., HILLS, N.J., TURNER, A.B., & CHEW, J.W. 2003. Measurement and Analysis of Ingestion Through a Turbine Rim Seal. *ASME Journal of Turbomachinery*, **125**, 505–512.
- GERMAIN, T., NAGEL, M., RAAB, I., SCHÜPBACH, P., ABHARI, R.S., & ROSE, M. 2010. Improving Efficiency of a High Work Turbine Using Nonaxisymmetric Endwalls—Part I: Endwall Design and Performance. *ASME Journal of Turbomachinery*, **132**, 021007 1–9.
- GIER, J., SABINE, A., REINMÖLLER, U., & NIEHUIS, R. 2002. Improving 3D Flow Characteristics in a Multistage LP Turbine by Means of Endwall Contouring and Airfoil Design Modification – Part 2: Numerical Simulation and Analysis. *Proceedings of the ASME Turbo Expo, Paper No. GT-2002-30353*.
- GONZALEZ, P., LANTERO, M., & OLABARRIA, V. 2006. Low Pressure Turbine Design for Rolls-Royce Trent 900 Turbofan. *Proceedings of the ASME Turbo Expo, Paper No. GT2006-90997*.

- GREGORY-SMITH, D., BAGSHAW, D., INGRAM, G., & STOKES, M. 2008. Using Profiled Endwalls, Blade Lean and Leading Edge Extensions to Minimise Secondary Flow. *Proceedings of the ASME Turbo Expo*, Paper No. GT2008-50811.
- GREGORY-SMITH, D.G. 1997. *Secondary and Tip Clearance Flows in Axial Turbines - Physics of Secondary Flows*. Von Karman Institute for Fluid Dynamics, Belgium.
- GREGORY-SMITH, D.G., INGRAM, G., JAYARAMAN, P., HARVEY, N.W., & ROSE, M.G. 2001. Non-axisymmetric Turbine End Wall Profiling. *IMEchE Journal of Power and Energy*, **215**, 721–734.
- GUSTAFSON, R., MAHMOOD, G., & ACHARYA, S. 2007. Aerodynamic Measurements in a Linear Turbine Blade Passage with Three-Dimensional Endwall Contouring. *Proceedings of the ASME Turbo Expo*, Paper No. GT2007-28073.
- HALL, S. 2012. *Contributions to High-Speed Linear Turbine Cascade Testing: Design of a Wind Tunnel Control System, and Investigation of the Effects of Compressibility on Total Pressure Losses*. Master's Thesis, Carleton University, Ottawa, Canada.
- HAN, W., YANG, Q.-H., HUANG, H.-Y., & JIA, L. 2002. Topology and Vortex Structures of Turbine Cascade with Different Tip Clearances. *Chinese Journal of Aeronautics*, **15**, 18–26.
- HARLAND, J.C., GREGORY-SMITH, D.G., HARVEY, N.W., & ROSE, M.G. 2000. Non-axisymmetric Turbine End Wall Design: Part II—Experimental Validation. *ASME Journal of Turbomachinery*, **122**, 286–293.
- HARRISON, S. 1990. The Influence of Blade Lean on Turbine Losses. *ASME Paper No.90-GT-55*.

- HARTLAND, J.C., GREGORY-SMITH, D.G., & ROSE, M.G. 1998. Non-Axisymmetric Endwall Profiling in a Turbine Rotor Blade. *ASME Paper No. 98-GT-525*.
- HARVEY, N.W., ROSE, M.G., TAYLOR, M.D., SHAHPAR, S., HARTLAND, J., & GREGORY-SMITH, D.G. 2000. Nonaxisymmetric Turbine End Wall Design: Part I—Three-Dimensional Linear Design System. *ASME Journal of Turbomachinery*, **122**, 278–285.
- HARVEY, N.W., BRENNAN, G., NEWMAN, D.A., & ROSE, M.G. 2002. Improving Turbine Efficiency Using Non-Axisymmetric End Walls: Validation in the Multi-Row Environment and with Low Aspect Ratio Blading. *Proceedings of the ASME Turbo Expo, Paper No. GT-2002-30337*.
- HASELBACH, F., SCHIFFER, H.P., HORSMAN, M., DRESSEN, S., HARVEY, N., & READ, S. 2002. The Application of Ultra-High Lift Blading in the BR715 LP Turbine. *ASME Journal of Turbomachinery*, **124**, 45–51.
- HERMANSON, K.S., & THOLE, K.A. 2000. Effect of Mach Number on Secondary Flow Characteristics. *International Journal of Turbo and Jet Engines*, **17**, 179–196.
- HODSON, H.P., & R.G., DOMINY. 1987. Three-Dimensional Flow in a Low-Pressure Turbine Cascade at Its Design Condition. *ASME Journal of Turbomachinery*, **109**, 177–185.
- HOEGER, M., CARDAMONE, P., & FOTTNER, L. 2002. Influence of Endwall Contouring on the Transonic Flow in a Compressor Blade. *Proceedings of the ASME Turbo Expo, Paper No. GT-2002-30440*.
- HUNTER, S.D., & MANWARING, S.R. 2000. Endwall Cavity Flow Effects on Gaspath Aerodynamics in an Axial Turbine: Part I - Experimental and Numerical Investigation. *Proceedings of the ASME Turbo Expo, Paper No. 2000-GT-651*.

- INGRAM, G., GREGORY-SMITH, D., & HARVEY, N. 2005. Investigation of a Novel Secondary Flow Feature in a Turbine Cascade With End Wall Profiling. *ASME Journal of Turbomachinery*, **127**, 209–214.
- ISLAM, A.M.T. 1999. *An Experimental and Computational Study of the Aerodynamics of Turbine Blades with Damage*. Doctoral Thesis, Carleton University, Ottawa, Canada.
- JEFFRIES, M. 2000. *Initial Investigation of Transonic Turbine Aerodynamics using the Carleton University High-Speed Wind Tunnel*. Doctoral Thesis, Carleton University, Ottawa, Canada.
- JOHANSEN, E.S., REDINIOTIS, O.K., & JONES, G. 2001. The Compressible Calibration of Miniature Multi-Hole Probes. *ASME Journal of Fluids Engineering*, **123**, 128–138.
- JOUINI, D.B.M. 2000. *Experimental Investigation of Two Transonic Linear Turbine Cascades at Off-Design Conditions*. Doctoral Thesis, Carleton University, Ottawa, Canada.
- KACKER, S.C., & OKAPUU, U. 1982. A Mean Line Prediction Method for Axial Flow Turbine Efficiency. *ASME Journal of Engineering for Power*, **104**, 111–119.
- KANDEBO, STANLEY W. 2004. GP7200 Readied for Tests. *Aviation Week and Space Technology*, **160**, 45–46.
- KNEZEVICI, D.C. 2011. *Controlling Secondary Flows in Very Highly-Loaded Low-Pressure Turbine Cascades*. Doctoral Thesis, Carleton University, Ottawa, Canada.
- KNEZEVICI, D.C., SJOLANDER, S.A., PRAISNER, T.J., ALLEN-BRADLEY, E., & GROVER, E.A. 2010. Measurements of Secondary Losses in a Turbine Cascade With the Implementation of Nonaxisymmetric Endwall Contouring. *ASME Journal of Turbomachinery*, **132**, 011013 1–10.

- KOBAYASHI, N., MATSUMATO, M., & SHIZUYA, M. 1984. An Experimental Investigation of a Gas Turbine Cooling System. *ASME Journal of Engineering for Gas Turbines and Power*, **106**, 136–141.
- KOPPER, F.C., MILANO, R., & VANCO, M. 1981. Experimental Investigation of Endwall Profiling in a Turbine Vane Cascade. *AIAA Journal*, **19**, 1033–1040.
- KOST, F., & NICKLAS, M. 2001. Film-Cooled Turbine Endwall in a Transonic Flow Field: Part I—Aerodynamic Measurements. *ASME Journal of Turbomachinery*, **123**, 709–719.
- LANGSTON, L.S. 2001. Secondary Flows in Axial Turbines – A Review. *Anal. of the New York Academy of Sciences*, **934**, 11–26.
- MACISAAC, G. 2011. *The Influence of Simulated Purge Flow on The Secondary Flow in a Low-Speed Linear Cascade*. Doctoral Thesis, Carleton University, Ottawa, Canada. In progress.
- MACISAAC, G.D., & SJOLANDER, S.A. 2013. Effects of Simplified Platform Overlap and Cavity Geometry on the Endwall Flow: Measurements and Computations in a Low-Speed Linear Turbine Cascade. *Proceedings of the ASME Turbo Expo*, Paper No. GT2013-95670.
- MARINI, R., & GIRGIS, S. 2007. The Effect of Blade Leading Edge Platform Shape on Upstream Disk Cavity to Mainstream Flow Interaction of a High-Pressure Turbine Stage. *Proceedings of the ASME Turbo Expo*, Paper No. GT2007-27429.
- MATERIAL SAFETY DATA SHEET. 2007 (December). *AX-13-5 Rocket Red Pigment*.
- MCLEAN, C., CAMCI, C., & GLEZER, B. 2001a. Mainstream Aerodynamic Effects Due to Wheel-space Coolant Injection in a High-Pressure Turbine Stage: Part I—Aerodynamic

- Measurements in the Stationary Frame. *ASME Journal of Turbomachinery*, **123**, 687–696.
- MCLEAN, C., CAMCI, C., & GLEZER, B. 2001b. Mainstream Aerodynamic Effects Due to Wheelspace Coolant Injection in a High-Pressure Turbine Stage: Part II—Aerodynamic Measurements in the Rotational Frame. *ASME Journal of Turbomachinery*, **123**, 697–703.
- MOBARAK, A., KHALAFALLAH, M.G., OSMAN, A.M., & HEIKAL, H.A. 1988. Experimental Investigation of Secondary Flow and Mixing Downstream of Straight Turbine Cascades. *ASME Journal of Turbomachinery*, **110**, 497–503.
- MOFFAT, R.J. 1982. Contributions to the Theory of Single-Sample Uncertainty Analysis. *ASME Journal of Fluids Engineering*, **104**, 250–260.
- MORRIS, A.W.H., & HOARE, R.G. 1975. Secondary Loss Measurements in a Cascade of Turbine Blades with Meridional Wall Profiling. *ASME Paper No. 75-WA/GT-13*.
- MOUSTAPHA, S.H., & WILLIAMSON, R.G. 1986. Effect of Two Endwall Contours on the Performance of an Annular Nozzle Cascade. *AIAA Journal*, **24**, 1524–1530.
- MOUSTAPHA, S.H., CARSCALLEN, W.E., & MCGEACHY, J.D. 1993. Aerodynamic Performance of a Transonic Low Aspect Ratio Turbine Nozzle. *ASME Journal of Turbomachinery*, **115**, 400–408.
- NAGEL, M.G., & BAIER, R.-D. 2005. Experimentally Verified Numerical Optimization of a Three-Dimensional Parametrized Turbine Vane With Nonaxisymmetric End Walls. *ASME Journal of Turbomachinery*, **127**, 380–387.

- PANCHAL, K., ABRAHAM, S., EKKAD, S.V., NG, W., BROWN, B.J., & MALANDRA, A. 2011. Investigation of Effect of End Wall Contouring Methods on a Transonic Turbine Blade Passage. *Proceedings of the ASME Turbo Expo, Paper No. GT2011-45192*.
- PANIAGUA, G., DÉNOS, R., & ALMEIDA, S. 2004. Effect of the Hub Endwall Cavity Flow on the Flow-Field of a Transonic High-Pressure Turbine. *ASME Journal of Turbomachinery*, **126**, 578–586.
- PASINATO, H.D., SQUIRES, K.D., & ROY, R.P. 2004. Measurements and Modeling of the Flow and Heat Transfer in a Contoured Vane-Endwall Passage. *International Journal of Heat and Mass Transfer*, **47**, 5685–5702.
- PERDICHIZZI, A. 1990. Mach Number Effects on Secondary Flow Development Downstream of a Turbine Cascade. *ASME Journal of Turbomachinery*, **112**, 643–651.
- PERDICHIZZI, A., & DOSSENA, V. 1993. Incidence Angle and Pitch-Chord Effects on Secondary Flows Downstream of a Turbine Cascade. *ASME Journal of Turbomachinery*, **115**, 383–391.
- POPOVIC, I. 2005. *Measured Steady and Unsteady Aerodynamic Performance of a Family of Three Highly-Loaded Low-Pressure Turbine Cascades*. Master's Thesis, Carleton University, Ottawa, Canada.
- POPOVIĆ, I., & HODSON, H.P. 2010. Aerothermal Impact of The Interaction Between Hub Leakage and Mainstream Flows in Highly-Loaded HP Turbine Blades. *Proceedings of the ASME Turbo Expo, Paper No. GT2010-22311*.
- PRAISNER, T.J. 2005. *Personal Communication*.

- PRAISNER, T.J., ALLEN-BRADLEY, E., & GROVER, E.A. 2007. Application of Non-Axisymmetric Endwall Contouring to Conventional and High-Lift Turbine Airfoils. *Proceedings of the ASME Turbo Expo*, Paper No. GT2007-27579.
- PRASAD, A. 2005. Calculation of the Mixed-Out State in Turbomachine Flows. *ASME Journal of Turbomachinery*, **127**, 564–572.
- REID, K., DENTON, J., PULLAN, G., CURTIS, E., & LONGLEY, J. 2006. The Effect of Stator-Rotor Hub Sealing Flow on the Mainstream Aerodynamics of a Turbine. *Proceedings of the ASME Turbo Expo*, Paper No. GT2006-90838.
- ROSE, M.G. 1994. Non-Axisymmetric Endwall Profiling in the HP NGV's of an Axial Flow Gas Turbine. *ASME Paper No. 94-GT-249*.
- ROSE, M.G., HARVEY, N.W., SEAMAN, P., NEWMAN, D.A., & MCMANUS, D. 2001. Improving the Efficiency of the Trent 500 HP Turbine Using Non-Axisymmetric End Walls. Part II: Experimental Validation. *Proceedings of the ASME Turbo Expo*, Paper No. 2001-GT-0505.
- SAHA, A.K., & ACHARYA, S. 2006. Computations of Turbulent Flow and Heat Transfer Through a Three-Dimensional Non-Axisymmetric Blade Passage. *Proceedings of the ASME Turbo Expo*, Paper No. GT2006-90390.
- SARAVANAMUTTOO, H.I.H., ROGERS, G.F.C, COHEN, H., & STRAZNICKY, P.V. 2009. *Gas Turbine Theory*. 6th edn. Prentice Hall, Toronto.
- SAUER, H., MÜLLER, R., & VOGELER, K. 2001. Reduction of Secondary Flow Losses in Turbine Cascades by Leading Edge Modifications at the Endwall. *ASME Journal of Turbomachinery*, **123**, 207–213.

- SCHUEPBACH, P., ROSE, M., GIER, J., RAAB, I., GERMAIN, T., & ABHARI, R. 2009. Non-Axisymmetric End Wall Profiles Including Fillet Radii, in a 1.5 Stage Axial Flow Turbine. *Proceedings of the 8th European Conference on Turbomachinery, Paper No. A159*.
- SCHUEPBACH, P., ABHARI, R.S., ROSE, M.G., & GIER, J. 2010. Sensitivity of Turbine Efficiency and Flow Structures to Varying Purge Flow. *Journal of Propulsion and Power*, **26**, 46–56.
- SCHUEPBACH, P., ABHARI, R.S., ROSE, M.G., & GIER, J. 2011. Influence of Rim Seal Purge Flow on the Performance of an Endwall-Profiled Axial Turbine. *ASME Journal of Turbomachinery*, **133**, 021011 1–10.
- SCHULER, P., KURZ, W., DULLENKOPF, K., & BAUER, H.-J. 2010. The Influence of Different Rim Seal Geometries on Hot-Gas Ingestion and Total Pressure Loss in a Low-Pressure Turbine. *Proceedings of the ASME Turbo Expo, Paper No. GT2010-22205*.
- SCHULER, P., DULLENKOPF, K., & BAUER, H.-J. 2011. Investigation of the Influence of Different Rim Seal Geometries in a Low-Pressure Turbine. *Proceedings of the ASME Turbo Expo, Paper No. GT2011-45682*.
- SIEVERDING, C.H. 1985. Recent Progress in the Understanding of Basic Aspects of Secondary Flows in Turbine Blade Passages. *ASME Journal of Engineering for Gas Turbines and Power*, **107**, 248–257.
- SIEVERDING, C.H., & WILPUTTE, PH. 1981. Influence of Mach Number and End Wall Cooling on Secondary Flows in a Straight Nozzle Cascade. *ASME Journal of Engineering for Power*, **103**, 257–264.
- SJOLANDER, S.A. 1975. *The Endwall Boundary Layer in an Annular Cascade of Turbine Nozzle Guide Vanes*. Master's Thesis, Carleton University, Ottawa, Canada.

- SNEDDEN, G., DUNN, D., INGRAM, G., & GREGORY-SMITH, D. 2009. The Application of Non-axisymmetric Endwall Contouring in a Single Stage, Rotating Turbine. *Proceedings of the ASME Turbo Expo, Paper No. GT2009-59169*.
- SONODA, T., HASENJÄGER, M., ARIMA, T., & SENDHOFF, B. 2007. Effect of Endwall Contouring on Performance of Ultra-Low Aspect Ratio Transonic Turbine Inlet Guide Vanes. *Proceedings of the ASME Turbo Expo, Paper No. GT2007-28210*.
- TAREMI, F. 2011. *Three-dimensional Flows in Transonic Axial Turbine Cascades*. Doctoral Thesis, Carleton University, Ottawa, Canada. In Progress.
- TAREMI, F., SJOLANDER, S.A., & ABO EL ELLA, H.M. 2008. Extending the Capabilities of a High Speed Wind Tunnel to Three Dimensional Flow Measurements in Transonic Turbine Cascades. *Proceedings of the 44th Joint Propulsion Conference, Paper No. AIAA-2008-5272*.
- TAREMI, F., SJOLANDER, S.A., & PRAISNER, T.J. 2010a. Measurements of Endwall Flows in Transonic Linear Turbine Cascades: Part I – Low Flow Turning. *Proceedings of the ASME Turbo Expo, Paper No. GT2010-22759*.
- TAREMI, F., SJOLANDER, S.A., & PRAISNER, T.J. 2010b. Measurements of Endwall Flows in Transonic Linear Turbine Cascades: Part II – High Flow Turning. *Proceedings of the ASME Turbo Expo, Paper No. GT2010-22760*.
- TAREMI, F., SJOLANDER, S.A., & PRAISNER, T.J. 2011. Application of Endwall Contouring to Transonic Turbine Cascades: Experimental Measurements at Design Conditions. *Proceedings of the ASME Turbo Expo, Paper No. GT2011-46511*.
- TAYLOR, J.R. 1997. *An Introduction to Error Analysis: The Study of Uncertainties in Physical Measurements*. 2nd edn. University Science Books, Sausalito.

- TORRE, D., VÁZQUEZ, R., DE LA ROSA BLANCO, E., & HODSON, H.P. 2011. A New Alternative for Reduction in Secondary Flows in Low Pressure Turbines. *ASME Journal of Turbomachinery*, **133**, 011029 1–10.
- WANG, H.P., OLSON, S.J., GOLDSTEIN, R.J., & ECKERT, E. 1997. Flow Visualization in Linear Turbine Cascade of High Performance Turbine Blades. *ASME Journal of Turbomachinery*, **119**, 1–8.
- WEBER, S., & HACKENBERG, H. 2007. GP7000: MTU Aero Engines Contribution in a Successful Partnership. *Proceedings of the AIAA International Conference on Air-breathing Engines, Paper No. ISABE-2007-1283*.
- WEISS, A.P., & FOTTNER, L. 1995. The Influence of Load Distribution on Secondary Flow in Straight Turbine Cascades. *ASME Journal of Turbomachinery*, **117**, 133–141.
- WU, P.-S., & ZHONG, D.-S. 2003. Flow Visualization for the Endwall Region of a First Stage Vane with a Forward Facing Entrance Step. *Proceedings of the PSFVIP-4, Paper No. F4012*.
- YARAS, M.I. 1990. *Measurement and Prediction of Tip-Clearance Effects in a Linear Turbine Cascade*. Doctoral Thesis, Carleton University, Ottawa, Canada.
- YASA, T., PANIAGUA, G., FRIDH, J., & VOGT, D. 2010. Performance of a Nozzle Guide Vane in Subsonic and Transonic Regimes Test in an Annular Sector. *Proceedings of the ASME Turbo Expo, Paper No. GT2010-22901*.
- ZESS, G.A., & THOLE, K.A. 2002. Computational Design and Experimental Evaluation of Using a Leading Edge Fillet on a Gas Turbine Vane. *ASME Journal of Turbomachinery*, **124**, 167–175.

- ZILLIAC, G.G. 1993. Modelling, Calibration, and Error Analysis of Seven-Hole Pressure Probes. *Experiments in Fluids*, **14**, 104–120.
- ZLATINOV, M.B., TAN, C.S., MONTGOMERY, M., ISLAM, T., & SECO-SOLEY, M. 2011. Turbine Hub and Shroud Sealing Flow Loss Mechanisms. *Proceedings of the ASME Turbo Expo*, Paper No. GT2011-46718.

Seven-Hole Probe Calibration and Data Reduction

Prior to use in the wind tunnel, the probe was calibrated at Mach numbers, 0.48, 0.67, 0.77, and 0.95. Calibration was done over the angle range of $\pm 20^\circ$ in 2° increments in both pitch and yaw using the high speed probe calibration rig (discussed in Section 3.4). For each of the pitch and yaw angle positions (α and β), the seven port pressures of the pressure probe are used to calculate the following calibration coefficients:

$$C_\alpha = \frac{2(P_4 - P_1) + P_3 - P_6 - P_2 + P_5}{3(P_7 - \bar{P})} \quad (\text{A.1})$$

$$C_\beta = \frac{P_3 - P_6 + P_2 - P_5}{\sqrt{3}(P_7 - \bar{P})} \quad (\text{A.2})$$

$$C_o = \frac{P_7 - P_o}{P_7 - \bar{P}} \quad (\text{A.3})$$

$$C_q = \frac{P_7 - \bar{P}}{P_o - P_s} \quad (\text{A.4})$$

where

$$\bar{P} = \frac{P_1 + P_2 + P_3 + P_4 + P_5 + P_6}{6} \quad (\text{A.5})$$

These calibration coefficients are calculated for each of the four calibration Mach numbers (0.48, 0.67, 0.77, and 0.95), constituting the calibration tables for the seven-hole pressure probe. These coefficients can be plotted as surfaces (functions of α and β) for each of the Mach numbers.

Figure A.1: Mach 0.77 seven-hole probe calibration surfaces: C_α (top left), C_β (top right), C_o (bottom left), and C_q (bottom right).

Figure A.1 shows the calibration surfaces for Mach 0.77, where C_α and C_β are primarily functions of α and β respectively, and C_o and C_q correlate well with the resultant flow misalignment angle, indicated by ϕ on the corresponding parts of the figure. C_q shows an

asymmetric distribution; this is not uncommon, as was also observed by Knezevici (2011) for a very similar seven-hole pressure probe.

To characterize an unknown flow field, the calibration coefficient tables with the seven port pressures of the unknown field being measured, are used in an iterative solving procedure combined with an interpolating table look-up method to extract the total pressure, dynamic pressure, Mach number and the two flow angles from the seven measured port pressures. Isentropic flow, and ideal gas are assumed to obtain the values for all primitive flow variables $(M, P_o, P, T, \rho, V, u, v, w)$.

Pressure Transducers Calibration

The wind tunnel pressure measurements were obtained using 11 Druck PDC-22 pressure transducers, and one Omega PX613 pressure transducer. These transducers were also used to obtain measurements on the High Speed Probe Calibration Test Rig. Each of the transducers are connected to a dedicated channel on the data acquisition and control systems. Part number, range, and function of each transducer were given in Table 3.1. The transducers are typically calibrated twice a year using a Druck DPI 605 Pressure Calibrator. Additionally, daily checking of the transducers wind-off values was done prior to tunnel operation. The calibration results in a linear correlation relating the voltage output to measured pressure for each transducer. The calibration data used during the experiment work in this thesis is given in Figure B.1, and the linear correlation coefficients are summarized in Table B.1. Note that transducers connected to Channels 1-6,9, and 10 are differential transducers, and the y-intercept values have been adjusted accordingly.

The Druck DPI605 Pressure Calibrator itself was calibrated in May 2010 using a US Airforce Standards calibration method (33K6-4-427-1), traceable to an NIST (National Institute of Standards and Technology) standard. During normal course of operation, the calibrator does not require recalibration (Druck, 1994).

Figure B.1: Pressure transducers calibration data.

Table B.1: Pressure transducers calibration coefficients.

Channel No.	Slope	Y-intercept
1	1351	0.3695
2	674.06	0.4553
3	665.4	0.224
4	670.59	0.106
5	1296.2	-0.154
6	1016.2	0.5724
7	1334	1.112
8	1333.9	-0.993
9	1360.4	0.5279
10	1348.5	0.201
11	698.23	-0.844

Flow Visualization Video Results

Real time videos were first taken of the flow visualization during startup to determine if the transient behaviour at startup and shutdown would distort the real flow patterns.

Figure C.1 are image captures from the video taken during the different phases of the wind tunnel operation. From left to right, they are: (a) before startup, (b) during ramp-up, (c) at steady operating condition, and (d) after shutdown.

Figure C.1: Video image captures during different phases of wind tunnel operation.

It was found that the startup transient behaviour had a minimum effect on the steady state pattern. The main effect occurred during the shutdown phase, between Figures C.1(c) and (d), due to the rapid closing of the control valve. This rapid closing caused a smearing of the pattern at the low shear stress area on the suction side of the blade, and a small reversal of the flow pattern at the trailing edge of the blade, falsely suggesting a small flow separation at the trailing edge, as shown in Figure C.2. After three repetitions at the

same operating condition the pattern established during the 50 second steady condition (Figure C.1(c)) remained unchanged except for the small change at the trailing edge due to the shutdown phase. This was noted, and was taken into consideration when interpreting the flow patterns.

Figure C.2: Suction surface flow visualization showing false separation.

Mixed-out Loss Calculations

The flow at the measurement plane is spatially non-uniform and additional losses will be generated as the flow mixes to a uniform condition. The losses at this condition, where the flow is spatially uniform, are referred to as the mixed-out losses.

The mixed-out losses were calculated by employing a control volume analysis, applying the conservation of mass, momentum, and energy principles. The control volume encompassed one blade pitch, spanning from the endwall to midspan, starting from the measurement plane, to a hypothetical fully mixed-out plane as shown in Figure D.1. The analysis assumed constant area mixing, while neglecting shear forces. Thus, the calculation does not account for additional endwall loss production downstream of the measurement plane. The calculation is similar to the method presented by Prasad (2005).

For the control volume and variables shown in Figure D.1, the equations and procedures used to obtain the mixed-out loss are presented below:

Area of measurement plane,

$$A = \frac{1}{2}Hs \quad (\text{D.1})$$

Figure D.1: Mixed-out loss calculation - control volume and variables.

Conservation of mass,

$$\dot{m} = \int \rho u \, dA = \rho_{mix} u_{mix} A \quad (D.2)$$

Conservation of momentum in x-direction,

$$F_x = \int (\rho u) u \, dA + \int P \, dA = \dot{m} u_{mix} + P_{mix} A \quad (D.3)$$

Conservation of momentum in y-direction,

$$F_y = \int (\rho u) v \, dA = \dot{m} v_{mix} \quad (D.4)$$

Conservation of momentum in z-direction,

$$F_z = \int (\rho u) w \, dA = \dot{m} w_{mix} \quad (D.5)$$

Conservation of energy,

$$\begin{aligned} \dot{E} = \frac{\gamma R}{\gamma - 1} \int (\rho u) T \, dA + \frac{1}{2} \int (\rho u) (u^2 + v^2 + w^2) \, dA = \\ \frac{\gamma \dot{m} P_{mix}}{(\gamma - 1) \rho_{mix}} + \frac{\dot{m}}{2} (u_{mix}^2 + v_{mix}^2 + w_{mix}^2) \end{aligned} \quad (D.6)$$

Ideal gas law,

$$\rho_{mix} = \frac{P_{mix}}{RT_{mix}} \quad (D.7)$$

The six equations D.2 to D.7 form a closed system, and can be solved to obtain the six unknowns at the mixed-out plane, u_{mix} , v_{mix} , w_{mix} , ρ_{mix} , P_{mix} , and T_{mix} .

This is accomplished by rearrangement and substitution of equations D.2 to D.6 to eliminate all unknowns except for the mixed-out static pressure, P_{mix} . This yields the following quadratic equation:

$$QP_{mix}^2 + LP_{mix} + G = 0 \quad (D.8)$$

where Q , L , and G are given by,

$$Q = \frac{1}{\dot{m}^2} \left(A^2 - \frac{2\gamma}{\gamma - 1} A^2 \right) \quad (D.9a)$$

$$L = \frac{2}{\dot{m}^2} \left(\frac{\gamma}{\gamma - 1} F_x A - F_x A \right) \quad (D.9b)$$

$$G = \frac{1}{\dot{m}^2} (F_x^2 + F_y^2 + F_z^2) - \frac{2\dot{E}}{\dot{m}} \quad (D.9c)$$

The solution for Equation D.10 is given by:

$$P_{mix} = \frac{-L - \sqrt{L^2 - 4QG}}{2Q} \quad (D.10)$$

where the negative root associated with the subsonic solution has been selected. Once P_{mix} is calculated, u_{mix} can be calculated from Equation D.3, and then ρ_{mix} can be calculated from Equation D.2. Equations D.4, and D.5 can be used to calculate v_{mix} , w_{mix} respectively. With P_{mix} , and ρ_{mix} now calculated, Equation D.7 can be used to obtain the mixed out static temperature, T_{mix} .

To obtain the stagnation temperature ($T_{o,mix}$), and pressure ($P_{o,mix}$) at the mixed-out plane, the following equations are used:

$$T_{o,mix} = \frac{\gamma - 1}{\gamma R} \frac{\dot{E}}{\dot{m}} \quad (\text{D.11})$$

$$P_{o,mix} = P_{mix} \left(\frac{T_{o,mix}}{T_{mix}} \right)^{\frac{\gamma}{\gamma-1}} \quad (\text{D.12})$$

Finally, using the mass-averaged inlet total pressure (P''_{o1}), the mixed-out stagnation pressure loss coefficient is calculated by:

$$Y_{mixed-out} = \frac{P''_{o1} - P_{o,mix}}{P_{o,mix} - P_{mix}} \quad (\text{D.13})$$