Blade - Vortex Interactions in High Pressure Steam Turbines

by

Venkata Siva Prasad Chaluvadi
Girton College

A dissertation submitted for the
degree of Doctor of Philosophy

Department of Engineering
University of Cambridge

November 2000

SUMMARY

A detailed experimental and numerical investigation of the transport of streamwise (passage) vortices in high-pressure axial turbines and their interaction with the downstream blade rows was performed. The results indicate large variations in the downstream flow field, notably the development of the secondary flows. The mechanism of passage vortex transport was studied in two differently configured high-pressure turbine stages. In the first configuration, the blades are radially stacked while the second configuration features three-dimensionally stacked high-pressure steam turbine blading.

The stator hub passage vortex is chopped by the downstream blade row in a similar way to the wake. The bowed vortex tube near the inlet to the rotor appeared to develop two counter-rotating legs extending back to the leading edges of the adjacent blades. These were termed the suction side leg and the pressure side leg. The two legs of the incoming passage vortex then convect with the respective velocities on the blade surfaces. The results are discussed for the radially stacked turbine and the 3-D turbine separately.

Radially Stacked Turbine:

The kinematic interaction between the stator and the rotor passage vortices has two effects. Firstly, the suction side leg of the stator passage vortex was displaced radially upwards over the developing passage vortex of the rotor blade. Secondly, the pressure side leg of the stator passage vortex was entrained into the rotor passage vortex.

The presence of stator viscous flow features in the rotor significantly enhanced the rotor secondary flow. The unsteady losses were estimated from unsteady numerical simulations to be 33% of the stage loss.

Three dimensionally Stacked Turbine:

The radial pressure gradient on the pressure surface resulted in the pressure leg of the stator passage vortex to migrate to a higher radial location from the action of image vortices inside the walls. The suction leg of the stator passage vortex is entrained in the rotor passage vortex at the hub. The lower velocities on the pressure surface resulted in the lower mass flow being effected by the pressure leg of the stator passage vortex. All these effects translated to lower interaction losses compared to a radially stacked case.

Delta wing + Three dimensionally Stacked Turbine:

The interaction of the streamwise vortices on the downstream blade row has been investigated by shedding only streamwise vortices into the downstream blade row. The loss measurements at the exit of the stator blade showed a significant increase in stagnation pressure loss due to the delta wing vortex transport. The increase in loss was 25% of the datum stator loss.

Flow underturning near the hub and overturning towards the midspan was observed at the exit of the rotor and it was showed that this phenomenon was a function of the incoming vortex strength at rotor inlet.

A method to calculate the unsteady loss generation in the blade row was developed from numerical simulations. An analytical model was developed using the concepts of the kinematic vortex transport inside the downstream blade. The analytical model brought out the significant effects of the vortex interaction with the downstream blade row very well.

Preface

The research in this dissertation was carried out at the Whittle Laboratory, Cambridge University Engineering Department, between October 1996 and November 2000. Except where specifically stated to the contrary, this dissertation is the result of my own work and includes nothing, which is the outcome of work done in collaboration. No part of the work has been submitted to any other university or place of learning.

I would like thank my supervisor, Professor Howard P Hodson, for making this work possible. His continuing guidance and support throughout this research is greatly appreciated. I would also like to thank Professor J D Denton for his constructive advice on several occasions. Particular thanks are also to Dr. Anestis I Kalfas and Dr. Liping Xu for the helpful advice and discussions.

I am grateful to many members of the Whittle Laboratory, past and present for their help and encouragement. I would like to thank Mr. Brian W Taylor and Mr. J Saunders for their assistance in the experimental set-up. I thank Mr. B. A. Leishman for proof-reading some of the chapters.

I am grateful for the support provided by Mitsubishi Heavy Industries, Japan. Particular thanks are to Mr. E Watanabe for his constant encouragement and interest shown in the present research.

I would like to thank my family, particularly my parents, for their encouragement and contributions over the years. Finally and above all, I thank my wife, Purnima, without whose patience, help and constant support this work would not have been possible.

Venkata Siva Prasad Chaluvadi
November 2000

Contents

Nomenclature

A, B	Kings law constants
C	Chord, Kings law constant
C_d	Dissipation coefficient
C_p	Static pressure coefficient, specific heat ratio
d	Diameter
$\hat{e}_{mean}$	Unit vector in the mean flow direction
E	Probe response in Volts
$E(\kappa)$	Power spectral density based on wave number
f	Blade rotating frequency
$\bar{f}$	Reduced frequency parameter
h	Blade span, enthalpy
H	Boundary layer shape factor (δ^*/θ)
I,Q,R	Second order polynomial functions of λ
k	Coefficient of thermal conductivity, turbulence kinetic energy
K	Probe calibration coefficient
l	Length
L_u	Turbulence length scale
$\dot{m}$	Mass flow rate
M	Mach number
n	Ratio of axial gap to axial chord, number of realisation
$\vec{n}$	Unit normal vector
N	Total number of realisations, Blade number
P	Pressure, blade pitch
psd	Power spectral density
r	Radius
R	Universal gas constant, outer irrotational core radius
Re_θ	Reynolds number based on momentum thickness
rms	Root mean square
rpm	Revolutions per minute
s	Specific entropy, streamwise distance

S'	Blade throat
$\dot{S}$	Entropy generation rate
$\dot{S}_a$	Entropy creation rate per unit surface area
$\dot{S}'''$	Entropy generation rate per unit volume
S_r	Ratio of vortex stretching
SF	Smoothing factor for numerical simulations
t	Time from a datum point
T	Temperature, Wake passing period, Torque
Tu	Turbulence intensity
U	Blade speed
V	Velocity
$\vec{V}$	Velocity vector
V_d	Velocity deficit in the wake centreline
V_{s0}	Mean flow velocity outside the wake
V_δ	Velocity at the edge of the boundary layer
W	Velocity of the fluid
x	Axial distance
Y	Stagnation pressure loss coefficient

Greek Symbols

σ	Entropy production rate per unit volume
α	Flow quantity, absolute yaw angle
β	Pitch angle
χ	Angle between velocity vector and the hot-wire sensor normal
δ	Vortex inner core radius
δ^*	Boundary layer displacement thickness
ε	Fractional change in air temperature from ambient conditions, turbulence dissipation rate
ϕ	Flow coefficient
γ	Hot-wire slant angle, ratio of specific heat, vortex filament angle
ω	Loss coefficient
λ	Hot-wire probe pitch angle relative to probe stem
Ω	Overheat ratio of the hot-wire sensor, angular velocity

θ	Boundary layer momentum thickness, flow angle
τ	Time for one wake passing period, time lag
Γ	Blade circulation
ξ	Yaw angle
η	Efficiency, Kolmogorov's microscale of turbulence
ζ	Probe yaw angle, vorticity
ρ	Density
∞	Free-stream
κ	Wave number
ν	Kinematic viscosity

Subscripts

A,B,C	of wire A, B, C
aru	additional random unsteadiness
avg	average
c	corrected
d	dynamic pressure
dyn	dynamic head
eff	effective
exit	stage exit
hub	hub
in	inlet
m	measured, mid-span
max	maximum
min	minimum
o,0	stagnation
p	pitch
r	radial, relative frame of reference
ref	reference
rep	representative
rms	root-mean-square
s	static, steady
sensor	sensor
sw	streamwise

t	total pressure
tip	tip
u	unsteady
v	total velocity
x	axial
y	yaw
θ	Tangential
1	blade row inlet
2	blade row exit

Superscripts

'	perturbation of the quantity
$\bar{}$	time mean of the quantity

Tradenames

Biodata	Registered name of Biodata Ltd.
Dantec	Trade name of Dantec Ltd.
Digiplan	Trade name of Parker Hannifin plc.
Druck	Trade name of Druck Ltd.
Kulite	Trade name of Kulite plc.
Microlink	Registered name of Biodata Ltd.
Scanivalve	Registered name of Scanivalve corp.
Zoc	Trade name of Scanivalve corp.

Introduction

1

1.1 Introduction

Sir Frank Whittle and Hans von Ohain were credited with being the first to use gas turbines for aircraft propulsion. Independently and without knowledge of each others activity, they developed the aircraft gas turbine. Their test engines ran and produced thrust for the first time in the spring 1937. After 1945, aircraft gas turbine development efforts have been directed towards increasing pressure ratios, turbine inlet temperatures, component efficiencies, bypass ratios, reliability and durability. As a result, the specific fuel consumption of the turbomachine has been reduced and thrust to weight ratios have increased. The turbomachine is now one of the worlds most important prime movers. The first jet engine developed only a few hundred pounds of thrust, while the latest generation of engines exceed 100,000 pounds thrust and the engines for the land-based power plants exceed 250MW in power output. Figure 1.1 shows the cross section of a modern aircraft gas turbine.

The development of turbomachinery is aimed at further improvements in efficiency, coupled together with an increase in the ratio of power output to weight and size, especially in aircraft applications. Efficiency is probably the most important performance parameter for turbomachines. This is especially true for gas turbine engines, whether used for aircraft propulsion or for land-based power plants, because their net power output is the difference between the turbine work and the compressor work. These are roughly in the ratio of 2:1 (this ratio depends on engine pressure ratio and turbine entry temperature), so a small change in the efficiency of either component causes a much larger proportional change in the power output. Over the years considerable effort has been expended in trying to improve the efficiency of all types of turbomachines and for many large machines the component efficiencies are now over 90 percent. This makes further improvements ever more difficult to achieve. Present levels of efficiency have been achieved by an ever-improving understanding of the fluid mechanics and thermodynamics of the flow, made possible by improved experimental and theoretical (especially computational) methods applied both to entire machines and to the individual components.

The lost efficiency in a turbine is in general caused by the irreversibilities associated with viscous flow phenomena, heat transfer across finite temperature differences and other non-equilibrium processes such as shock waves. Contrary to external aerodynamics, in turbomachinery flows, viscous effects are not restricted to boundary layers but are also present in the freestream due to the spanwise and the pitchwise gradients of the flow. The viscous effects are also present within free shear layers and in the dissipation and mixing of viscous flow structures.

In the past, the design of axial-flow machines was based on the experimental evidence of a large number of cascade tests. In recent years, viscous 2D and 3D steady flow prediction methods have become available, thus reducing the need for cascade testing. The designer, in using the results of steady flow analyses or cascade tests, assumes that the blade rows of an axial flow turbomachine are sufficiently far apart, that the flow is steady in both the stationary and rotating frames of reference. From the testing of completed designs, it has become apparent that the efficiency of an axial flow turbine stage is less than predicted by this approach. Clearly, there are differences between the flows which occur in rotating machines and their equivalent cascades. One of these differences is that the flow in a real machine is unsteady from the relative movements of the adjacent blade rows. The flows are also three-dimensional due to the circumferential as well as spanwise gradients in total pressure and temperature induced by upstream blade rows, combustors and inlet systems. In addition, the upstream blade wakes and vortices while convecting through the downstream blade rows undergo pressure and velocity changes continually so that they mix in an unsteady environment that is different from that modelled in traditional cascade tests. The blade wakes arise from blade surface boundary layers while the streamwise vortices occur when the flow with inlet vorticity is turned by a cascade due to the distortion of the inlet endwall boundary layer vortex filaments.

The requirement for the aero-engine manufacturers to produce lighter and more powerful gas turbines has led to high pressure turbines with highly loaded, low aspect ratio blades and smaller inter blade row gaps. These design factors have increased the overall spatial and temporal complexities of the turbine flow field. Thus, the effects of the blade wake and streamwise vortices interaction with downstream blade row have become even more important.

Due to the lack of the realistic models for loss generation in the unsteady flow environment, designers rely on the use of experience factors for steady state loss correlations to account for these unsteady effects (Dunham (1996)). However, these factors do not necessarily reflect the true physical nature of the loss generation mechanisms in the unsteady environment. Improved analytical models of the loss generation mechanisms in unsteady flow would provide increased understanding of the turbine flow field and would lead to improved gas turbine performance.

A significant amount of research activity has recently been directed towards understanding the effect of unsteadiness on turbine performance. One of the major sources of unsteadiness was found to be the interaction between the streamwise vortices with the downstream blade row (Binder (1985), Binder *et. al.* (1986), Sharma *et. al.* (1988)). These vortices include passage vortices that are developed because of the endwall boundary layers, the vortices that form from leakage flows and the scrapping effects of the rotor, and the trailing edge vortices caused by a variation of spanwise circulation. These streamwise vortices are shown to have a major influence on the secondary flow and viscous flow behavior of the downstream blade row (Sharma *et. al.* (1990)). There is paucity of investigations in the literature that are directed towards understanding the interaction of these upstream vortices on the flow characteristics of the downstream airfoil rows and practically none quantify the effect of these vortices interactions on additional loss generation. The present study attempts to address some of these issues.

1.2 Research Objectives

This thesis examines the interaction of the upstream viscous flow features on the performance of the downstream blade row. This investigation has the following main objectives:

(1) Identify the unsteady phenomena of significance in high-pressure turbines.

(2) Understand the interaction of the streamwise vortices with the downstream blade row. This includes understanding the transport mechanism of the passage vortices inside the downstream blade row and the mixing of the passage vortices unsteadily in the downstream blade row.

(3) Quantify the effect of upstream streamwise vortices interaction on the loss of the downstream blade row.

(4) Translate this knowledge into design recommendations.

These objectives were met by a comprehensive experimental testing and numerical simulation programme. The interaction of the stator passage vortices with the rotor blade was studied in two different turbine stages. The first corresponded to a radially stacked high-pressure gas turbine stage, while the second corresponded to a three dimensionally stacked and swept high-pressure steam turbine stage. The interaction of the streamwise vortices on the downstream blade row has been investigated by shedding only the streamwise vortices into the downstream blade row. The following section presents the organisation of the thesis.

1.3 Thesis Outline

General concepts in unsteady flows including the origins of unsteadiness and its effect on the performance are discussed in chapter 2. This chapter also reviews the work carried out by the other researchers on secondary vortex interactions with the downstream blade row.

Chapter 3 describes the experimental apparatus, instrumentation and various measurement techniques used in the present investigation. It also describes the steady and unsteady computations used in this study.

Chapter 4 presents the results of the study on vortex - blade interaction in a typical radially stacked high-pressure gas turbine stage. It describes the three-dimensional flow field within the blade rows with the help of flow visualisation experiments, steady and unsteady flow field measurements and CFD simulations. It proposes a simple kinematic model for the transport of secondary passage vortices through the downstream blade row.

Chapter 5 presents the results of the study on the interaction of stator passage vortices with the downstream rotor blade row in a three dimensionally stacked and swept high-pressure steam turbine stage representative of the modern design methodology. The steady and unsteady three-dimensional flow field in the turbine stage is discussed using surface flow visualisation, surface pressures, exit flow field measurements and CFD calculations. This chapter presents a method of identifying and quantifying the loss generated in various regions of the blade using the data from numerical simulations. The effect of stator secondary flow on the development of rotor secondary flow is also illustrated.

To isolate the effect of passage vortex interaction with the downstream blade row, a method was devised to simulate the passage vortex with a half delta wing. The characteristics of the delta wing vortex with downstream axial distance were investigated in a wind tunnel and reported in Chapter 6. Chapter 6 also presents the effect of rotating half-delta wing vortex interaction with the downstream stator and rotor blade in the low-speed research turbine. The additional loss generated from the vortex interaction with the downstream blade was evaluated both experimentally and numerically.

Chapter 7 discusses the numerical and analytical modeling of the vortex – blade transport in the downstream blade row. Finally, a discussion of the current work, the main conclusions and suggestions for future work are presented in Chapter 8.

2 Literature Review

2.1 Introduction

The aim of this chapter is to provide an overview of the published work that is relevant to the current investigation. Given the scope of the present research, a comprehensive review of all of the contributing research is not possible. With a brief discussion of the loss generation mechanisms and secondary flow development in turbine blades, the present chapter focuses on the detailed unsteady phenomena in turbomachines. This chapter discusses the sources of unsteady flow, unsteady loss generation mechanisms and provides an insight into the mechanisms of interaction between the stator and the rotor of a modern high-pressure turbine. Unsteady flow calculations form an important part of the current investigation. A brief summary of the steady and unsteady multi blade row prediction methods is given in this chapter. This chapter also discusses briefly the use of lean and sweep in designing modern steam turbine blades and reviews the analytical models for the wake-blade and vortex-blade interactions in turbomachinery.

2.2 Loss Mechanisms

The principal aim of the current investigation is to achieve a greater level of understanding of the complex flows in high-pressure turbines with particular emphasis on loss generating mechanisms in unsteady flow. Denton (1993) gave an extensive review of the loss generating mechanisms in turbomachinery. He stressed the importance of the physical understanding of the origins of loss within the turbomachinery flows and argued against the continued use of non-physical correlations in the design process. He identified three principal sources of loss in a turbomachinery environment:

(a) Viscous shear in boundary layers, shear layers and mixing processes

(b) Non-equilibrium processes such as shock waves

(c) Heat transfer across the finite temperature differences

Entropy is chosen as the most suitable measure of irreversibility or loss in the present investigation because its value is independent of the frame of reference and it is a convected quantity. Entropy, which is created during an irreversible process, may be compared to smoke as it diffuses into the surrounding fluid and convected downstream. These sources of loss generation are described briefly in the following section.

2.2.1 Losses in Boundary Layers

Boundary layers are regions of steep velocity gradients and large shear stresses. These highly viscous regions are responsible for much of the loss created in a turbomachine, with a high proportion of this loss being created in the inner part of the boundary layer where the velocity gradients are the steepest as shown by Dawes (1990a). Denton (1993) derived an expression for the entropy production rate per unit surface area '$\dot{S}_a$' in a two-dimensional boundary layer and showed it to be a strong function of the velocity at the edge of the boundary layer 'V_δ'. The entropy production rate can be non-dimensionalised to give a dissipation coefficient 'C_d' and is given as

$$C_d = \frac{T\dot{S}_a}{\rho V_\delta^{\,3}} \tag{2.1}$$

The value of C_d depends upon the state of the boundary layer and the Reynolds number based on the local boundary layer thickness. Figure 2.1 compares the dissipation coefficients for laminar and turbulent boundary layers following the work of Schlichting (1966). For turbulent boundary layers, the value of dissipation coefficient does not vary greatly. The large difference in dissipation coefficients for laminar and turbulent boundary layers with Reynolds numbers in the range $300 < Re_\theta < 1000$ highlights the importance of transition prediction in the assessment of loss production in the turbomachinery boundary layers. A large amount of research is currently being undertaken in the field of transition prediction with particular emphasis on the effect of unsteady wake passing on the transition process. Some of this work is discussed in more detail in section 2.5.2.

2.2.2 Losses in shear flows and mixing processes

Regions of steep velocity gradients such as wakes, vortices, separated shear layers and leakage jets are responsible for a large amount of entropy production. This is partly a consequence of the high rates of shear but largely a result of the high levels of turbulent viscosity present in these viscous flow regimes. The exact level of entropy produced by the viscous shear forces acting to mix out these flow features depends on the environment in which mixing occurs. Denton (1993) showed that a wake mixing out in an accelerating flow would produce less entropy than mixing out in a diffusing flow. Conversely, the application of Kelvin's theorem to the streamwise vorticity in a vortex shows that the vortex stretching will increase the secondary kinetic energy, thereby increasing the loss as the vortex mixes out. In a turbomachine, the wakes and the vortices formed by one blade row are rarely fully mixed out before the next blade row is reached. Therefore, some of the mixing will occur in an unsteady environment. The implications of this unsteadiness for the production of entropy are poorly understood (Denton (1993)). As the unsteady loss generating mechanisms have significant bearing on the present research, the available literature on unsteady flows is discussed in more detail in the following sections.

2.3 Unsteady flows

Flow in turbine blade rows is highly unsteady (with the exception of the first stator blade), because they periodically encounter flow distortions generated by the upstream blade rows and combustors. This unsteadiness has important consequences for the turbine stage efficiency, blade loading, mechanical fatigue, heat transfer, thermal fatigue and noise generation. The induced unsteady flow depends upon the scale of the upstream disturbances like streamwise vortices, wakes and temperature streaks. These unsteady flow-generating factors can be classified based on the physical mechanisms involved as

(a) Potential interactions of the upstream and the downstream blade rows.
(b) Interactions of the upstream wakes with the downstream blade rows.
(c) Shock wave interaction with the downstream blade rows.
(d) Leakage and secondary flow vortices interaction with the downstream blades.

Each of these interactions has their own zone of influence. It is useful to characterise the degree of unsteadiness by evaluating the reduced frequency parameter as defined by Lighthill (1954). The reduced frequency is the ratio of time taken by the given particle for convection through the blade passage to the time taken for the rotor to sweep past one stator passage. It is expressed as

$$\bar{f} = \frac{f\,x}{V_x} = \frac{\text{Convection time}}{\text{Distrubence time}} \tag{2.2}$$

The magnitude of the reduced frequency is a measure of the relative importance of unsteady effects compared to quasi-steady effects. If $\bar{f} >> 1$, unsteady effects are significant, when $\bar{f} \approx 1$, unsteady and quasi-steady effects coexist. The reduced frequency $\bar{f}$ also represents the number of wakes (or other stator features) found in a single rotor passage at any instant in time. Although the reduced frequency parameter characterises the unsteadiness in a qualitative way, it does not shed much light on the magnitude or the interplay of these effects. For this, the interactions themselves have to be considered. These unsteady loss-generating mechanisms are discussed in the following sections in some detail.

2.4 Potential Interactions

Potential interactions arise because all the blades have circulation and therefore a potential field, which propagates, throughout the space. When one blade row is moving relative to the next, the pressure field established by one blade row interacts with the potential field of the other. The magnitude of this effect depends on a number of factors. Parker and Watson (1972) gave series of relationships for unsteady pressure and velocity fluctuations for a two dimensional cascade. The potential field associated with a blade row propagates both upstream and downstream of the blade row and it varies approximately in proportion to the quantity

$$\exp\left\{-2\pi\sqrt{1-M^2}\,\frac{x}{p}\right\} \qquad \text{for values of } M < 1.0 \qquad (2.3)$$

where ‘x’ is the axial distance from the blade row, ‘p’ is the pitch of the blade row and ‘M’ is the local Mach number. Note that the decay scales with the blade pitch and not the blade chord. This is important if different blade loadings (pitch-chord ratios) are being considered. While the equation represents an oversimplification of the actual situation, it does serve to show that at low Mach numbers, the axial distance from the blade row has more bearing on unsteady pressure than the inlet Mach number. However, as the sonic condition is approached, the decay becomes much less. This means that in high Mach number flows, potential interactions will tend to be stronger than at lower speeds. If the Mach number is high enough, then the potential field will propagate without decay.

Confirmation of the (relative) weakness of the propagation of downstream potential fields, can be found in many experiments in literature. Parker and Watson (1972) confirmed that the much earlier observations of Kemp and Sears (1955) were applicable to blades in cascades and suggested the effects of potential interactions will be insignificant for axial spacing greater than about 30% of the blade pitch. Walker and Oliver (1972) demonstrated the separate existence of the potential and wake effects in an axial flow compressor. Their measurements clearly demonstrated the rapid decay of the potential influence when compared to that of the wake. Another effect of potential interaction is due to the changes in flow incidence it brings to the downstream blade row, which in turn result in unsteady blade circulation and vortex shedding as reported by Kemp and Sears (1955).

In the turbine under investigation the flow is subsonic (turbine exit Mach number is 0.13) with a minimum blade row axial gap of approximately 25% blade pitch. This short discussion of potential interactions suggests that these interactions will be weak in the present investigation. Although these effects should not be dismissed entirely, it is shown below that wake-blade interactions dominate the generation of unsteadiness in the flow.

2.5 Wake – Blade Interaction

The potential influence of a blade row extends upstream and downstream and decays exponentially as discussed in the previous section, whereas a blade wake is only convected downstream. A wake profile can be characterised by a relative velocity deficit, typically 10-30% of the free stream value, wake width and turbulence intensity. The values depend primarily on how far behind the blade row the measurement is made. Raj and Lakshminarayana (1976) investigated the development of the blade wakes shed from an isolated compressor rotor. They gave wake velocity decay rate as

$$\frac{V_d}{V_{s0}} = \exp\left\{-\frac{\pi^2}{R_s}\left(\frac{s}{S'}+\frac{s_0}{S'}\right)\right\} \tag{2.4}$$

Where ‘s’ is the streamwise distance from the blade trailing edge, ‘S'’ is blade throat, ‘V_d’ is the velocity deficit in the wake centreline and V_{s0} is the mean flow velocity outside the wake. They have identified two different regions of interest: near wake and far wake region. For the near wake region, the two constants in the equation 2.4 are given as R_s=0.71 and s_0/S'=0.013 and for the far-wake region the constants are given as 14.00 and 3.46 respectively. The decay rate of the wakes was observed to be more rapid in the near-wake and less rapid in the far-wake regions than the decay rates of wakes shed from isolated airfoils or rectilinear cascades. They have also found that the turbulence intensity decays much slower than the velocity deficit in the wake.

The velocity deficit in a upstream blade wake can be perceived as an incidence variation by the downstream blade row, while the turbulence inside the wake can change the nature of the boundary layer from laminar to turbulent, generating additional losses. It can also be shown that the influence of a wake will persist through several blade rows even if they are spaced far apart. Other researchers like Schlichting (1987) have shown that for a 2D steady wake, the velocity deficit is known to decay as $(s)^{-1/2}$. However in a multistage environment, the wake mixing could be somewhat different. The laser measurements of Stauter *et al*. (1991) in a two stage axial flow compressor showed that the rotor wake decay to be proportional to $e^{-2.295n}$, where 'n' is the ratio of the axial gap to the axial chord and the magnitude is about 1/4 to 1/2.

Kerrebrock and Mikolajczak (1970) illustrated the effect of upstream velocity variations due to wakes and hot streaks with velocity triangles as shown in figure 2.2. In some flow situations, especially for the blade row downstream of a combustor, high velocity jets exist due to large circumferential gradients in temperatures and are generally referred as hot-streaks. Figure 2.2 shows that the low velocity fluid has a normal velocity component towards the suction side of the downstream blade indicating that the high turbulence, low-velocity fluid from the upstream airfoil wake will migrate towards the suction side of the airfoil. In a similar manner, high velocity fluid will migrate towards the pressure side of the downstream airfoil. This migration of the fluid particles has the following major effects on the downstream blade row:

(i) Alterations in the boundary layer characteristics of the airfoil through its effect on the transition process.
(ii) Effects the secondary flow generation for the downstream passages.
(iii) Effects the wake mixing loss due to wake stretching or compression.

These effects are discussed in detail in the following sections.

2.5.1 Wake transport in the downstream blade row

One of the first studies of the interaction of wake with blades was conducted by Meyer (1958). Using thin airfoil theory and drawing upon the work of Kemp and Sears (1955), he presented a solution for the interaction of a upstream blade wake with the moving downstream blade row as illustrated in figure 2.3. Each wake is initially represented as a perturbation of the uniform flow. The wakes are transported with the main flow and are chopped into segments by the downstream blade row. Inside the blade passage, the wake continues to behave as negative-jet. The velocity induced by the negative jet causes a build up of the wake fluid on the suction surface and a removal of the wake fluid from the pressure surface.

There are considerable differences between the flow over an un-cambered airfoil and that through a turbomachinery blade row. Smith (1966) noted that as a result of the blade circulation, the wake segments are sheared and stretched as they progress through the blade passage. These phenomena are particularly evident in axial turbines. The experimental results of Hodson(1985a) and the subsequent numerical results of Hodson(1985b), Giles(1987), Rai(1987), Korakianitis(1991), Dawes(1993) and Hodson and Dawes (1996) confirm that, a simple kinematic theory can be used to explain the movement of wakes through the downstream blade rows.

The convection of a wake through the downstream blade row calculated from the numerical simulations by Hodson and Dawes (1996) is presented in figure 2.4. The wake before entering the blade passage undergoes ‘bowing’ due to the higher velocities in the middle of the passage compared to near the blade surfaces. It also experiences ‘shearing’ near the suction surface and ‘stretching’ near the pressure surface because the part adjacent to the suction surface convects more rapidly than the part adjacent to the pressure surface. The net result of the bowing, stretching, shearing and distortion is that the wake appears to be concentrated on the suction surface at blade row exit with a tail stretching back to the rotor leading edge as shown in figure 2.4. This kinematic wake transport also effects the loss generated from mixing of these wakes in the downstream blade row as shown by Smith(1966), Adamczyk(1996), Deregel and Tan (1996), Valkov(1995), Van de Wall *et al.* (2000). Smith (1966) showed that when the axial gap between the rotor and the stator reduced from 37% chord to 7% chord a one-percent increase in efficiency was obtained in an axial compressor.

The total loss associated with the blade wake can be taken as the loss generated behind the blade row from the blade boundary layers and the loss that would occur if the blade wake was allowed to mix-out with the surroundings in isolation by viscosity alone. This is never the case in turbomachinery as the blade rows are close to each other most of the time to reduce the engine length. If the blade wake can be thought of as two vortex sheets, then the vorticity kinematics begins to play a role in changing the decay of the wake. The wake mixing in the downstream blade row now involves two mechanisms, viscosity as well as the kinematics of the wake stretching, shearing. Van de Wall *et al.* (2000) has shown that the vorticity kinematics can act to reduce (or increase) the velocity deficit associated with a wake from Kelvin's theorem resulting in reducing (or increasing) the mixing losses associated with the wake.

2.5.2 Effect of unsteady transition

The periodically changing upstream flow can have a large effect on the development of the downstream blade boundary layers. The subject of wake induced boundary layer transition is a large area of research by itself and a complete review of the available literature is beyond the scope of this thesis. A comprehensive review of transition in turbomachinery components is given by Mayle(1991). He lists the four modes of transition and describes the mechanism by which they occur:

(i) *Natural transition* – amplification of Tollmien-Schlichting instability waves in low free stream turbulence

(ii) *Bypass transition* - high free stream turbulence level

(iii) *Separated flow transition* – laminar separation bubbles

(iv) *Periodic unsteady transition* – wake impingement (a special case of unsteady bypass transition)

The influence of wake induced unsteadiness on the development of boundary layers has been investigated by Pfeil *et al.*(1982) on a flat plate with upstream rotating bars. They found that the impingement of the wakes on the flat plate forces early transition of the boundary layer from laminar to turbulent. Hodson (1983) measured detailed boundary layer velocity profiles on a rotor blade in both a stationary cascade and a rotating rig environment. He showed that the profile loss of a rotor blade at mid span was 50% higher than the same blade profile loss tested in a cascade with steady inflow.

As the wake convects through the blade passage, the suction surface boundary layer is influenced by the patches of increased turbulence and perturbations of pressure and velocity. Assuming the boundary layer to be of single state (i.e. either laminar or turbulent but not transitional), the time-mean properties of the boundary layer will be largely unaffected due to this turbulence as suggested by Lighthill (1954). The same can not be said of the effect of wakes on the transition of a boundary layer from laminar to a turbulent state. Schulte (1995) has shown that it is possible to reduce loss generated by a highly loaded low pressure turbine blades at low Reynolds numbers (typically of the order 1.3 x 10^5), if they are subjected to incoming wakes. This loss reduction is due to the periodic elimination of the laminar separation bubbles in the diffusing region of the airfoil. By exploring this feature, one can effectively increase the blade loading by as much as 20% above the current blade designs for the same loss as shown by Howell (1999).

In addition to affecting the characteristics of the blade boundary layers, wakes from upstream blade row also affect the generation of secondary flows as shown by Sharma *et al.* (1985, 1988) and Herbert and Tiederman (1989). Sharma *et al.* (1985) described the measurements obtained by using fast response pressure probes in a large scale-rotating rig. They have shown that the hub secondary flow vortex varies from a distinct structure to diffused structure and becomes non-existent at a later time during one stator passing period. This indicates that the secondary flow generation mechanisms are strongly influenced by the upstream circumferential distortions such as wakes. Sharma *et al*. (1988) observed that the periodic variation in the size and the strength of the secondary flow vortices in this experimental investigation resulted in almost 40% variation in secondary flow losses for the rotor passage. The influence of the unsteady flow on the secondary flow development of the downstream blade row is further discussed in section 2.7.

2.6 Shock Wave – Blade Interaction

Compression is seldom a desirable feature of turbines, however, transonic turbines are commonly used to obtain high-stage pressure ratios and so shock waves do occur. The most serious consequence of having a transonic flow in a turbine is a shock system at the trailing edge, as illustrated in figure 2.5. The low base pressure formed immediately behind the trailing edge can generate a very large trailing edge loss. The flow expands around the trailing edge to this low pressure and is then re-compressed by a strong shock wave at the point where the suction and pressure side flows meet. Denton (1993) suggested that there is entropy generation from the intense viscous dissipation at the edges of the separated region immediately behind the trailing edge and also from the strong shock at the end of this region. The interaction between shocks and boundary layers can lead to unsteady boundary layer separations and an increase in loss for transonic velocities as shown by Atkin and Square (1992) and Scherbakov *et al.* (1997). As the present investigation is restricted to low speed research this shock-boundary layer interaction is not a source of unsteadiness and will not be discussed further.

2.7 Vortex – Blade Interactions

Harrison (1989) has shown that in low aspect ratio machines, the secondary flows in the form of streamwise vortices are significant across the blade covering up to 30% of the blade span. These vortices are convected downstream towards the next blade row where they interact with the main flow. Although this form of interaction can be significant, the phenomenon is not fully understood and a comprehensive review is yet to be published. As it has significant bearing on the present research, the available literature is discussed in more detail in the following sections. Before discussing the streamwise vortices interaction with the downstream blade row, a brief description of how these streamwise vortices (secondary flow vortex and tip leakage vortex) are created and their effects on the flow is given below.

2.7.1 Secondary flow vortex

The secondary flow in a blade row can be defined as any flow, which is not in the direction of the primary or streamwise flow. Figure 2.6(a) shows the classical secondary flow model of Hawthorne (1955). This model shows how secondary flows occur when the flow with inlet vorticity is turned by a cascade due to the distortion of the inlet boundary layer vortex filaments. A comprehensive introduction to classical secondary flow prediction is given by Gregory-Smith (1984).

A physical model describing the development of a secondary flow is depicted in figure 2.6(b). Review papers by Sieverding (1984) and Wang (1995) provide comprehensive summaries of the research on secondary flow structure and outline the most significant developments. The illustrated model exaggerates the rate of rotation of the vortices. The main passage vortex, for example, only undergoes a total of ¼ rotation within the blade passage and a total of about ½ rotation to a plane one chord downstream of the cascade.

Sharma and Butler (1987) have shown that the main parameters that influence the strength of secondary flows are blade turning, pitch to chord ratio, aspect ratio and inlet vorticity. The main consequence of the secondary flow is an increase in loss near endwalls and a radially non-uniform distribution of exit flow angle. This vortex gives rise to an area of overturning close to the endwall and an area of underturning nearer the mid-span. Most experimental studies have been performed with collateral endwall boundary layers, which have only a normal component of vorticity. In a turbomachine, the endwalls upstream of a blade row are, in general, rotating relative to the blade row in question and the inlet boundary layer is skewed. The results from the experiments of Bindon (1980), Boletis *et al.* (1983), Walsh and Gregory-Smith(1987) show that skew has a significant effect on the development and migration of the passage vortex and loss core. The direction of rotation in a turbine means that the streamwise vorticity introduced by the skewed inlet boundary layer strengthens the streamwise vorticity at the exit. Boletis *et al.* (1983) observed that with a skew representative of a machine environment, the hub passage vortex was convected beyond mid-span, counter to the radial pressure gradient and the results are also confirmed by Walsh and Gregory-Smith(1987).

2.7.2 Tip Leakage Vortex

The necessary clearance between the rotating and the stationary components within the turbine gives rise to a clearance flow and hence loss of efficiency. Blade sealing configurations fall into two main categories: unshrouded and shrouded blades. Pictorial representation of the tip leakage flow in shrouded and unshrouded blades is given in figure 2.7. The leakage flow over unshrouded blades occurs as a result of the pressure difference between the pressure and suction surfaces and is dominated by the trailing vortex shed near the blade tip. The flow produced by this vortex reduces the local turning performed by the blade and hence the work extracted from the flow. As a consequence of the viscous effects in the tip clearance gap, entropy is produced. The second major aspect is the subsequent mixing of the flow which has passed through the tip clearance gap with that of the main flow. The detailed structure of the flow within the tip region of a cascade has been extensively researched by Wadia and Booth (1982); Sjolander and Amrud (1987); Moore and Tilton (1988); Bindon (1989); Yaras and Sjolander (1990) and Heyes *et al.* (1992). Denton and Cumpsty (1987) gave a simple prediction model for the first order estimates of loss and reduced work due to tip clearance.

Comparatively very little published literature is available on leakage flows in shrouded blades. The pressure difference over the shroud provides the driving force for the fluid to pass into a shroud cavity and contract into a jet, downstream of a knife-edge seal as shown in figure 2.7. As the jet diffuses, some of its kinetic energy is dissipated in a turbulent mixing zone. Further downstream, the leakage flow is re-introduced back into the main flow where another mixing process occurs. Experimental studies have been limited to those of Denton and Johnson (1976), Lewis(1993) and Wallis(1997). Denton(1993) provided a simple prediction model for the leakage flow and loss in a shrouded blade.

The models for leakage loss in unshrouded blades neglect the relative motion between the blade tip and the endwall. In a compressor this relative motion increases the leakage flow and in a turbine it acts to reduce the leakage flow. This relative motion introduces scrapping flow and scrapping vortex near the leading surface of the blade. The importance of the relative motion has been investigated by Morphis and Bindon (1988), Yaras and Sjolander (1992). Similar studies are yet to be conducted in the tip gap of a shrouded turbine blade row.

2.7.3 Vortex – Blade Interaction

One of the first studies of the interaction of streamwise vortices with downstream blades was conducted by Binder (1985). He conducted the tests in a low aspect ratio, high hub-to-tip ratio transonic turbine stage leading to a strong secondary flow from the stator with non-uniform stator exit flow field characterised by high endwall losses and large passage vortices. Under such circumstances, the passage vortices persist and interacts with the downstream blade row. Binder (1985) took laser velocimeter measurements inside the rotor to study this interaction. He measured large increase in random unsteadiness in the front part of the rotor passage in the regions associated with the stator secondary flow. This was attributed to the breakdown of the vortex and proposed that the vortex breakdown occurred due to vortex filling and cutting and the strong deformation of the vortex cross-sectional area as the vortex enters into the rotor. The energy of the vortex motion is converted into random fluctuation energy during breakdown. Binder *et al.* (1986) proposed a more detailed mechanism for the vortex-rotor interaction. They suggested that shortly after the vortex is cut by the rotor blade, a disturbance is created which propagates along the vortex axis at the local speed of sound whilst simultaneously being swept downstream at the local convection velocity. They located the origin of the turbulence production to be close to the pressure-side stagnation region.

In another investigation in a one and half stage turbine, Sharma *et al*. (1985) have shown that the flow in the first rotor is highly unsteady and strongly influenced by the flow generated by the upstream stator. Two distinct rotor exit flow fields were identified corresponding to the maximum and minimum interaction of the rotor flow with the upstream stator wake and streamwise vortices. During the maximum interaction, the stator flow merged with the rotor wake, the mid channel flow was fairly uniform and two strong secondary flow vortices were present. During minimum interaction, the stator flow entered the rotor passages between the airfoils, the mid channel flow was non-uniform, and the secondary flow vortices were less defined. Sharma *et al*. (1985) suggested that the upstream features convect through the rotor passage without interacting with the rotor blades during the minimum interaction period and vice versa. The present author considers the converse to be more correct. The rotor exit flow is a result of the interaction between the rotor and upstream vane at all the times in one stator passing period.

From the same tests, Sharma *et al.* (1988) presented the spanwise distribution of the pitchwise averaged exit flow angle of the second stator and this compared with the rotor exit data as shown in figure 2.8. The data for the rotor shows overturning in the endwall regions and underturning in the midspan regions indicating the classical secondary flow. However, the data for the secondary stator shows underturning in the endwall regions and overturning in the mid-span regions, indicating that this flow is strongly influenced by the incoming secondary vortices of the rotor. A similar atypical distribution of flow angles in spanwise direction was also reported by Hobson and Johnson (1990) in his two-stage turbine. Significant overturning, together with an axial velocity deficit, was observed in their turbine downstream of the second stator at 30% blade span. This unsteady effect raised substantial interest, as it had not been observed in the tests carried out by the other researchers (Boletis and Sieverding (1991), Hodson *et. al.* (1993)). Although the configurations of Sharma *et al*. (1988) and Boletis and Sieverding (1991) were similar, the influence of the unsteady interactions on the steady performance was distinctly different and hence must be related to the strength of the incoming secondary flow of the turbines.

Very few experiments have been conducted on the interaction of the streamwise vortex with a downstream blade row. Reference has been made to the unpublished work of LaFleur *et al.* (1988) by Sharma *et al.* (1990). They introduced tip and hub vortices, generated by a stationary cascade of blades, at the inlet to the downstream turbine cascade in a water tunnel. The flow interactions were visualised by utilising dyes and laser lighting techniques at various axial planes in the downstream cascade. The incoming vortices simulate the quasi-steady effect of the secondary flow vortices generated by upstream blade rows in a multistage turbine. The results are shown in figure 2.9. The vortices introduced close to the suction surface of a turbine blade were found to convect along the surface in a stable manner hugging the airfoil suction surface. When these vortices were introduced on the pressure side, they were found to periodically collapse and reform in an unsteady manner. This result showed that even from a steady incident vortex, an unsteady flow field has developed. However, in reality, the vortex-blade interaction encountered in turbines is due to the relative motion between the upstream vortices and the downstream blade row. Hence the extent to which the conclusions from the above stationary incident vortex experiment are applicable to actual conditions is unclear.

Moving upstream vortex interaction with the downstream blade was investigated by Van de Wall *et al.* (1995). These tests were carried out in a water tunnel. The vortex interaction was visualised using a pH based flow visualisation technique and photographed using a video camera. Van de Wall *et al*. (1995) confirmed the conclusions of LaFleur *et al.* (1988) regarding the existence of unsteady flow field when stationary vortices were introduced on the pressure surface. In addition, they showed that the periodic collapsing and reforming of the vortices when they are introduced on the pressure surface is due to the helical breakdown of the incident vortex identical to that observed by Chanaud(1965). The oscillatory behaviour of the vortex occurred at a constant frequency. As the streamwise vortex approaches the adverse pressure gradient near the stagnation region, it is perturbed from the primary streamline. The vortex, which appears to oscillate, actually rotates around the initial incident streamline forming a three-dimensional helical pattern as shown in figure 2.10. This rotation of the helix is in addition to the rotation within the vortex core. The helical pattern of the vortex induces a velocity opposite to the initial velocity, which thus produces a stagnation point in the flow field on the axis of the initial streamline.

In another experiment, the upstream vortex generators were moved at a constant speed before the turbine cascade to simulate the preceding blade row. The unsteady vortex motion associated with breakdown was not confined to the stagnation region and the pressure surface, but occurred at all regions. The vortex breakdown was not dominated exclusively by the classical helical breakdown but also by the vortex cutting and vortex bursting as shown by Binder (1985). These tests are much closer to actual conditions, but they still give only a qualitative picture of the vortex transport and do not consider the effect of radial pressure gradient on the incoming vortex.

In another investigation in a one and half stage turbine, Walraevens *et al.* (1998) has shown that the vortices leaving the rotor passage strongly influenced the flow within and behind the second stator.

It is apparent from the above discussion that the incoming streamwise vortices have an important effect on the flow distribution for the downstream blade row. Needless to say these unsteady flow phenomena would not be expected to be predicted by the available steady state three-dimensional codes. A brief discussion on the multistage steady and unsteady blade row prediction methods is given section 2.8. It is also very important to model these unsteady wake-blade interaction and vortex-blade interaction using simple analytical methods for more physical understanding. A brief review of the existing analytical models in this area is discussed in the following section.

2.7.4 Analytical Modelling of Wake-Blade and Vortex-Blade Interaction

One of the first studies of secondary flow generation using analytical modelling was proposed by Hawthorne (1955) as shown in figure 2.11. He modelled the incoming boundary layer as a vortex filament *ab* and showed that as this vortex filament convects through the downstream blade row to *def*, it produces three forms of vorticity. These are identified in figure 2.6(a). The distributed secondary vorticity is a result of the turning of the inlet vortex filament while the trailing filament vorticity is generated due to the velocity differential between the suction and pressure surfaces. The trailing shed vorticity arises due to the spanwise variation of the blade circulation. He has also postulated the individual contribution of these mechanisms towards the overall secondary flow generation. Though this approach is idealised it provided an insight into the origins of secondary flow.

Similar models were developed for understanding the wake-blade interaction in axial flow compressors by Deregel and Tan (1996), Adamczyk (1996), Valkov (1995) and van de Wall *et al.* (2000). If the blade wake can be thought of as two vortex sheets, then vorticity kinematics can be used in understanding the wake interaction. By calculating the change in wake length from inlet to the exit of the downstream blade row, Valkov(1995) calculated the wake mixing loss. Deregel and Tan modelled the interaction based on control volume approach and kinematics of wake-blade interaction.

A secondary flow vortex from one blade row convects through a downstream blade row very much like a wake. However, the implications for the loss are very different. According to Kelvin's theorem, the circulation around a streamtube remains constant and so if the diameter of the tube is reduced by stretching, the streamwise vorticity is amplified. When a vortex is stretched (or compressed) longitudinally, it can be shown that its secondary kinetic energy will vary as the square of its length (Denton (1993)). Hence, stretching a vortex will greatly amplify its secondary kinetic energy and when this is subsequently dissipated by viscous effects to a uniform flow, it will generate additional loss. There is no simple model available in the literature for modelling vortex-blade interaction. An attempt will be made to model the vortex-blade interaction as a part of the present project.

2.8 Multi Blade Row Prediction Methods

Improved understanding into the development of the flow in a turbine environment can be obtained through the analysis of numerical results obtained from the steady and unsteady numerical simulations of the turbine stage. The relative motion between the blade rows is being modelled at present by one of two types of computational approaches.

The first method simulates the time averaged flow field within a blade passage with some modelling of the real flow to remove the effect of unsteadiness. This is normally achieved by the use of a mixing plane approximation at some axial distance between the blade rows (Denton and Singh (1979), Denton (1990), Dawes(1990b)) or by the inclusion of deterministic stresses as described by Adamcyzk(1985,1999). In the mixing plane approach, the stator/rotor interface plane matching conditions ensure the overall conservation of mass, momentum and energy across the interface. This approach assumes that the flow is mixed out at the 'mixing plane' to a flow with circumferentially uniform enthalpy and entropy before entering the next blade. This approximation could be erroneous if the blade rows were physically very close to each other, in which case the mixing would occur further downstream within the next blade row.

Adamcyzk (1985) attempted to model the gradual, unsteady mixing that occurs in practice by the addition of axisymmetric stress terms to the steady Navier-Stokes equations. Adamcyzk (1985) approximated these terms by solving the convection of a wake through an axisymmetric model of the downstream blade row. Although the deterministic stress model is undoubtedly more realistic than the mixing plane model it is still idealised. In particular, the real wake will collect on the suction surface rather than mix out axisymmetrically.

The second type of model involves the calculation of the unsteady flow field within the machine which is much more representative of the true environment. The complexity and the expense of obtaining high quality time-accurate experimental data has also given rise to a need for reliable unsteady calculations in order that a greater understanding of unsteady turbomachine aerodynamics can be achieved, although at present, unsteady calculations are computationally too expensive for routine use. As the speed and storage capacity of computer increase, unsteady calculations will form an important part of the design process and will help to improve performance and reduce development costs. Unsteady calculations form an important part of the current investigation. The following discussion summarises some of the work carried out in this field and highlights the main findings.

Hodson (1985b) developed a code to predict the unsteady wake-rotor interaction phenomena observed in his experimental investigation. He used a two-dimensional inviscid formulation based on that of Denton (1982). To remove the restriction of equal numbers of stator and rotor blades, a time-lagged periodic boundary condition was imposed in the blade-blade plane. In the absence of any real viscous forces, artificial viscosity inherent to the difference scheme was used to model the viscous decay of the wakes. He presented calculations that agree reasonably well with the measurements and concluded that the wake-rotor interaction is dominated by the inviscid effects.

Giles (1987) also developed a code for calculating the two-dimensional inviscid flow of the wake-rotor interaction. He solved the two-dimensional Euler equations using Lax-Wendroff method of Ni (1982). His results showed similar trends to those of Hodson (1985b). Giles (1990) extended his analysis to model stator-rotor interaction in a turbine. He found good agreement between his calculations and experimental results for the interaction of the stator trailing edge shocks with the rotor. The analysis was further extended to three dimensions by Saxer and Giles (1993).

Gallus *et al.* (1994) compared the experimental results from a single stage turbine with both steady and unsteady 3D Reynolds averaged Navier-Stokes calculations. They found that the unsteady interaction moved the stator hub secondary vortex away from the hub region and that unsteady calculations gave a better match with the experimental data than the steady calculations. Sharma *et al.* (1990), Ni and Sharma (1990) and Sharma *et al.* (1994) carried out similar investigations using both steady and unsteady codes. They found that the average loading was well predicted by the steady calculation even in the presence of periodic unsteadiness. Takanashi and Ni (1990) performed the calculations with hot streak into the stator inlet flow field. The unsteady code showed how the hot gases migrated to the pressure side of the rotor whilst the cooler gases migrated to the suction side. This phenomena was not predicted by the steady calculation. Further unsteady calculations also showed how stator vortices significantly affect the spanwise distribution of the rotor exit gas angles. They concluded that the unsteady codes are expected to make a significant contribution to the design of low aspect ratio turbines but further advancements in the speed of computing were necessary before unsteady codes could be used on routine scale. The unsteady numerical simulations were extensively used to understand the unsteady flow phenomena in the present investigation.

One of the main problems faced by the turbomachinery designer is the difficulty in quantifying the additional loss, which arises as a consequence of unsteady effects. Current design procedures often rely on experience factors based on the empirical data to account for the increased losses observed in the real turbine environment. The experimental measure of the unsteady loss is extremely difficult since much of the loss is generated in close proximity to solid surfaces and its evaluation requires accurate measurement of the instantaneous pressure and temperature fields. In order to overcome these experimental difficulties, some researchers have turned to CFD to try and provide an estimate of the unsteady loss. This also has its limitations since the quality of the answer depends upon the turbulence model and transition criteria employed. Nevertheless, some progress has been made in this area.

Dawes (1994) performed a three-dimensional unsteady Navier-Stokes calculation on a transonic compressor stage and compared the entropy production of the time-averaged flow to the time averaged entropy production. The difference between the two loss evaluation techniques was equal to 16% of the overall stage loss. He attributed this unsteady loss to unsteady shock motion in the rotor and the interaction of the rotor tip leakage vortex and the stator endwall flows. An attempt has been made to calculate the entropy production rate and unsteady loss from the unsteady numerical simulation results in the present investigation.

2.9 Three Dimensional Blade Design

The continual development of advanced axial turbines towards a higher degree of stage loading increases the three-dimensional flow phenomena. It has been observed that a considerable portion of efficiency debit is due to secondary flow not only in low aspect ratio high-pressure turbines but also in highly loaded low-pressure turbines. Many researchers have investigated the mechanism of the suppression of secondary flows by three-dimensional stacking and blade sweep both experimentally and analytically. The secondary flow phenomena are different in three dimensionally designed blades to radially stacked blade in a turbomachine. The former design philosophy emphasises the reduction of secondary flows compared to the latter. One of the objectives of the present investigation is to understand the phenomena of vortex-blade interaction in a high-pressure turbine. Therefore, it is important to understand the differences in transport phenomena in both the turbines designed using these philosophies. A brief review of the principles of three-dimensional design at this juncture is considered appropriate and given in the following section.

The effects of sweep have been discussed in detail by Potts (1987). Sweep induces stream surface twist because the spanwise component of velocity tends to remain constant as the flow passes through the blade row whilst the axial velocity is increased on the suction surface and decreased on the pressure surface. The angle of twist is greatest near the mid-span but its effect on blade loading is greatest near the endwalls. The result is that the swept forward leading edge is unloaded near the end-wall while the swept back leading edge experiences an increased loading. The converse is true at the trailing edge.

The effects of lean are even stronger than that of sweep. Hourmouziadis and Hubner (1985) concluded that the compound lean stacking reduces the secondary flows by reducing the blade to blade pressure gradient near the endwalls. The effect has been confirmed experimentally by Walker (1987) and Harrison (1989). The effect of blade lean can be understood by using the meridional streamline curvature equation (Walker (1987)). If the pressure side of the blade is leaned towards the hub, for example, producing a radially inward force on the fluid then the pressure level increases at the hub (and decreases near the casing). The change of pressure level near the end walls will force a corresponding change of velocity level. The reduced velocity levels near the hub lead to reduced loss levels. The blade with compound lean (i.e. the pressure side leaned towards both the hub and casing endwalls but broadly radial in mid-span) should permit better overall performance by unloading the blade near the lossy endwalls and loading the blade more in the supposedly more efficient mid-span. Denton (1994) described the advantages of designing in three dimensions. Further details of the present three-dimensionally designed stator and rotor blades are given in section 3.3.

2.10 Conclusions

The sources of unsteadiness in a turbine blade have been listed and the individual contributions of these sources to unsteadiness has been discussed. It has been shown that unsteady flows effect the performance of the turbine stage to a large extent. The streamwise vortices interaction with the downstream blade row resulted in large variations in the downstream flow field especially on the development of downstream secondary flow generation. A maximum of 40% variation in secondary flow losses with time is observed in the rotor passage over one stator passing period by Sharma *et al.* (1988). From this it can be concluded that this interaction is highly significant and there is a need for understanding the interaction. The passage vortices transport inside the downstream blade row is not well understood. Though some of the experiments were conducted to visualise the same, they are all restricted to water tunnel experiments in two-dimensional cascades at very low Mach numbers and Reynolds numbers. The effect of the radial pressure gradient, which exists in actual turbine environment, can not be represented in these tests.

Most of the studies in the literature are conducted in a multistage environment where various forms of secondary flows occur simultaneously in the blade row and difficulty is encountered in isolating a particular secondary flow, its cause and the effect. There is a paucity of investigations in the literature directed towards understanding the interaction of these upstream vortices on the flow characteristics of the downstream airfoil rows. Practically none quantify the effect of these vortex interactions in terms of additional loss generation. It is very important to know whether these interactions are significant in terms of unsteady loss generation as this will have a significant bearing in the design philosophy of the turbine.

These issues have been addressed in the present thesis. In the present study, a systematic investigation of the interaction of the vortices on the downstream blade row is carried out for understanding the phenomena of vortex interaction. The objectives of the present investigation are given in section 1.2. These objectives are met through a combination of experimental testing and numerical simulations. An attempt has also been made to calculate the entropy production rate from the unsteady numerical simulation results, and to calculate the contribution of various unsteady flow phenomena to the stage loss.

3 Experimental Setup and Methods

3.1 Introduction

During the course of the research described in this dissertation, various experimental and numerical investigations were undertaken on two low aspect ratio high-pressure axial flow turbines. The first corresponded to a radially stacked high-pressure gas turbine stage, while the second corresponded to a three dimensionally stacked and swept high-pressure steam turbine stage. The flow field in the turbines was investigated using both flow visualisation and steady and unsteady measurements. This chapter describes the test vehicles, the instrumentation and the measurement techniques used in performing the experiments. Steady and unsteady numerical simulations also formed an important part of the present investigation, and a brief description of these is also provided.

3.2 Test Facilities

Two of the test facilities at the Whittle Laboratory of Cambridge University were used for the experiments described in this thesis. The first was the large-scale low speed ‘Peregrine’ rotating turbine rig. The second was the low speed rectilinear ‘Duplex’ wind tunnel. In addition to these low speed rigs, a low speed calibration facility was used for the calibration of both pneumatic and hot-wire probes.

3.2.1 Low Speed Rotating Turbine Test Rig

The low speed rotating turbine test facility is shown in figure 3.1(a) and illustrated schematically in figure 3.1(b). The large inlet contraction and honeycomb screens are followed by a parallel section, which contains the single stage turbine. Air is drawn from the atmosphere through the turbine by a 400 hp centrifugal fan operating at a constant speed of 2100 rpm. The mass flow rate of the air is controlled by throttle vanes situated downstream of the turbine at the inlet to the fan. Flow enters the rig through a filter to remove dirt particles, then passes through a large inlet contraction fitted with honeycomb flow straighteners to remove the effects of ground vortex or other airborne disturbances. The power output from the turbine is absorbed by an eddy current dynamometer, which automatically regulates the speed of the turbine to within one rpm of a pre-set value.

The large scale of the rig made it possible to measure the flow field inside the blade passage and both upstream and downstream of the blade rows. The present work was carried out in high-pressure axial flow turbines with a casing diameter of 1.524m and a hub-tip ratio of 0.8. The rig is fitted with a slip ring system, which transfers power to and signals from the rotor mounted instruments. Four different configurations of axial flow turbine were tested in this facility. These are described in the later sections of this chapter.

3.2.2 Low speed Wind tunnel

A half delta wing fitted to a flat plate was used to generate a vortex. The distance between the delta wing and the leading edge of the flat plate was selected to represent the endwall boundary layer at the inlet to the turbine in the ‘Peregrine’ rig. A trip wire of 1.2mm diameter was used to ensure that the boundary layer on the flat plate was turbulent at the inlet to the delta wing. By varying the angle of incidence and the size of the delta wing, the circulation and the size of the vortex were altered. Figure 3.2 shows the geometry of the half delta wing used in the present investigation.

The test section used for the low speed experiments is schematically shown in figure 3.2. A variable speed centrifugal fan supplied the airflow and the fine adjustment of the throughput was achieved by throttling the inlet. The flow enters the test section through honeycomb flow straighteners followed by a large contraction, which reduces the boundary layer thickness. The test section was not rigidly fixed to the tunnel exit, thus avoiding the vibrations of the centrifugal fan being transferred to the test section and the probes fixed to it. Instead, the test section was fixed to the laboratory floor and the gap between the tunnel exit and the test section was sealed with foam draft excluder. A three axis traverse system was fixed to the test section wall to investigate the flow in both the pitchwise and spanwise directions at various axial locations.

3.3 Turbine test cases

Two turbines were tested in the ‘Peregrine’ test rig. The turbines are briefly described in this section. The second turbine was tested in three different configurations.

3.3.1 Turbine 1 – Conventional 2D high pressure turbine

Turbine 1 is representative of 2D-design philosophy used in gas turbine applications. The stage consists of 36 stator blades of 150mm chord and 42 rotor blades of 124mm chord with aspect ratios of 1.01 and 1.22 respectively. The blades are radially stacked from hub to casing. The rotor blades have 1.0 mm of clearance between the blade tip and the casing. Trip wires of 1.2 mm in diameter are used to ensure that the boundary layers at the hub ($\delta^*/h=0.006$, H=1.41) and casing ($\delta^*/h=0.0069$, H=1.42) are turbulent at the inlet to the stator row. These are located at two stator axial chords upstream of the stator blade row. Table 3.1 and figure 3.1(b) gives the further details of the turbine. Hodson (1985a) described the test facility in detail. The experimental data for this test case was acquired by Dr. R. Baneighbal. However all the processing and interpretation of the data that form a part of the present dissertation were performed by the present author.

3.3.2 Turbine 2 – 3D high pressure turbine

Turbine 2 is a high-pressure turbine representative of the 3D-design philosophy used in steam turbines. Mitsubishi Heavy Industries, Japan, carried out the aerodynamic design of the present blading. The blade design is aimed to produce a geometrical arrangement that can reduce the secondary flow losses. The resultant blade geometry is a combination of front loaded profiles at the mid-span and aft loaded profiles close to the endwalls. Three-dimensional stacking of the blades at the leading edge was given particular attention as it was considered to have a significant effect on the secondary flow growth.

The stage consists of 36 stator blades of 150mm chord and 42 rotor blades of 124mm chord with aspect ratios of 1.01 and 1.22 respectively. The blades were moulded from glass reinforced epoxy resin in a N.C. machined aluminium mould. These were sealed to both hub and casing using silicon rubber. Figures 3.3(a) and (b) show the stator and rotor blades respectively. The rotor blades are shrouded and have minimum axial and radial clearances of 1mm. The casing above the rotor shroud was formed as a two piece split ring. Sealing knife-edges are used over the rotating shroud in order to form a labyrinth seal arrangement. Figure 3.4 shows the rotor shroud arrangement. Sacrificial pieces made of balsa wood were placed against the rotating faces of the shroud in the axial direction. It is important that in the radial direction any possible contact between the shroud and the knife-edges to have no catastrophic consequences, and are therefore made out of different materials. As an additional precaution, an electronic circuit was devised which automatically shuts the rig when a metal to metal contact between the rotating and the stationary equipment is detected.

This circuit proved to be particularly useful during the commissioning phase of the present research programme. This is mainly because the rotor had to be balanced *in situ* in which case the vibrations will often overshoot the running clearances in the initial trails. In order to avoid leakage between the rotating and the stationary parts of the hub and the casing into the working section, a V-ring seal was used. A general view of the rotor at the time of assembly is shown in figure 3.5.

Trip wires of 1.2 mm in diameter were used to ensure that the boundary layers at the hub and casing are turbulent at the inlet to the stator row. These were located two stator axial chords upstream of the stator blade row. Further details of the turbine are given in Table 3.2. This test configuration is hereafter referred to as the 'Turbine 2 - Datum' configuration.

To simulate a rotating endwall before the stator at hub, an additional rotor was attached to the front of the turbine 2. The additional rotor had no blades. It consisted of a rotating hub ring, a stationary casing of the rotor. This configuration is hereafter referred to as the 'Rotating Hub' configuration.

The interaction of the streamwise vortices on the downstream blade row was investigated by shedding the streamwise vortices into the downstream blade row. The present turbine rig test configuration is schematically represented in figure 3.6. Half delta wings fitted to the rotating hub ahead of the stator were used to generate a row of vortices that were shed into the downstream stator blade row. The delta wings were fixed to the hub ring by means of a 3mm diameter high tensile screw. The gap between the bottom of the delta wing and the cylindrical hub surface was filled with RTV silicone rubber to avoid flow leakage. This test configuration henceforth is referred to as the 'Delta Wing' configuration.

3.4 Data Acquisition and Instrumentation

Details of the instrumentation used to study the flow field within the research stage are described in this section. The data acquisition can be divided into two broad categories: quantitative data acquisition and flow visualisation. The quantitative data acquisition involved the sampling of pressures, temperatures, velocities and torque using different types of transducers. The flow visualisation involved studying the patterns made by the three-dimensional flow on the bounding surfaces of the test turbine.

The acquisition of all the quantitative data was performed using a 233MHz Pentium II personal computer connected to a ‘Biodata Microlink’ unit through an IEE-488 interface. The Fortran data logging software controlled the data acquisition hardware through the ‘Microlink’ data bus. This ‘Microlink’ contained a 16-channel data acquisition card linked to a 12bit analogue to digital converter, which was used to sample and store the analogue voltages from the transducers to a resolution of 2.5mV.

For the measurement of pressures in the absolute frame of reference, two 48-port Scanivalve systems were used. Each Scanivalve unit incorporated a ± 3450 Pa Druck PDCR-22 differential pressure transducer. In the rotating frame a separate 48-port Scanivalve system mounted on the rig axis was used for the measurement of pressure. This Scanivalve unit also incorporated the same type of pressure transducer. Control of the port stepping and transducer voltage sampling was achieved through a slip-ring system for the relative frame measurements.

The ZOC module has eight separate differential pressure transducers rated at ± 8620 pa with integrated amplifiers and has the advantage that all pressures can be measured simultaneously. All the transducers were calibrated regularly using a ‘Druck’ DPI-520 pressure control system.

The temperatures were measured using T-type thermocouples referred to a cold junction whose temperature was measured using a platinum resistance thermometer.

The rotor speed was measured using an optical switch mounted on the rotor shaft coupled to a period timer, which measures the interval between successive once per rev pulses.

The radial and circumferential traversing of various probes used in the course of the experimental investigation was achieved using a three-axis closed loop traverse system. A stepper motor coupled with an optical encoder drives each of the three axes. A ‘Digiplan’ CD IFX stepper motor drive system connected to the PC through a serial port controlled the movement of the traverse gear.

3.5 Probes and Measurement techniques

During the course of the experiments, extensive measurements were taken using various probes and measurement techniques. Measurements in the wind tunnel were carried out with a five-hole pneumatic probe. Steady measurements in the test turbine were made using five-hole probes to resolve the main flow. Flattened Pitot tubes were used to measure the boundary layers. A single slanted hot-wire probe (SSHW) and a three-axis hot-wire probe were extensively used for unsteady measurements in the test turbine. The probes and the measurement techniques applied in the research are described briefly in this section.

3.5.1 Five -Hole probe

A five-hole probe with a diameter of 2.05 mm was used to measure the flow field downstream of the blade passage. It has a cone semi-angle of 45° and side pressure holes drilled perpendicular to the sides of the cone, as shown in figure 3.7. It also illustrates the convention used to number the 5-hole probe pressure tubes. The centre hole responds to the local stagnation pressure of the flow while the difference between the upper and lower holes is a measure of the pitch angle. Likewise, the difference in pressure between the side holes is a measure of the yaw angle. The pressure difference is not only a function of flow angle but also of the dynamic pressure of the flow. The difference between the true and measured values of the stagnation pressure is also dependent on these parameters. Thus, it is usual to calibrate the probe at an appropriate velocity level and then form the data into non-dimensional coefficients.

The calibration was performed over $\pm 30^0$ yaw and $\pm 30^0$ pitch to give a calibration map made up of 29 yaw angles and 29 pitch angles. Geometric nulling was used to define 0 degrees yaw and 0 degrees pitch both in the calibration and in the later measurement set-ups. Calibration experiments were performed at the velocities of 45 m/sec and 35 m/sec, which are typical velocities encountered in the area traverses downstream of the cascade. As the effect of Reynolds number was found to be very weak, only the calibration map obtained at the high Reynolds number was used in processing the experimental data. Calibration coefficients were chosen to give good resolution over most of the calibrated range. The coefficients were defined as follows:

$K_y = (P_2-P_3) / (P_{max}-P_{min})$ ------ Yaw angle coefficient (3.1)

$K_p = (P_4-P_5) / (P_{max}- P_{min})$ ------ Pitch angle coefficient (3.2)

$K_t = (P_o-P_1) / (P_{max}- P_{min})$ ------ Stagnation pressure loss coefficient (3.3)

$K_d = (P_o-P_s) / (P_{max}- P_{min})$ ------ Dynamic pressure coefficient (3.4)

Where $P_{max} = P_1$ (probe centre pressure) and P_{min} = Minimum of (P_2, P_3, P_4, P_5)

Figure 3.8 shows the sample calibration map of yaw and pitch coefficients with respect to yaw and pitch angles. To avoid proximity errors, area traversing was restricted to areas at least one probe diameter from the walls. Various extension lengths of the five-hole probes were used in the present research due to the geometric restrictions inside the stage. Some of the probes are of a modular design for easy assembly inside the test rig. Figure 3.9 show the various five-hole probes used in this research.

3.5.2 Single Slant Hot-Wire

To derive all three components of the unsteady velocity vector requires either a multi-wire probe or a single slanted wire that is positioned in at least three different orientations with respect to the flow. Each method has its advantages and disadvantages. The multi-wire probes give good resolution of the time-dependent fluctuations within the flow, but lacks in spatial resolution and suffer from wire and prong wake interference between the wires.

The hotwire inclined at 54.7^0 to the probe axis, when rotated about its probe support to three positions 120 degrees apart, keeps the sensing element mutually orthogonal to each other. This allows simpler data acquisition to derive the three components of the velocity vector, thus reducing the problems of data processing. Given the power of modern computers, this advantage hardly matters today. The unsteady flow is unlikely to be of constant direction and significant prong-wake interference can occur at some angles as the wake from the long prong spoils the flow onto the wire. Inaccuracies in the manufacture of the wire also make this type of data acquisition susceptible to errors.

For this experimental investigation, the unsteady velocity field downstream of the rotor trailing edge was measured using the single slant hot-wire technique described by Kuroumaru *et. al.* (1982) and developed to work on the low speed rotating compressor rig in the Whittle Laboratory by Goto (1991). The sensor used was a 'Dantec' hot-wire probe type 55P12 with a wire angle of 45 degrees. The hotwire was used in constant temperature mode with an overheat ratio of 1.8 and controlled by a 'Dantec' 56C17 bridge and 56N20 signal conditioner. Figure 3.10, shows a diagram of the slant hot-wire and defines the notation used in the velocity vector calculation method.

The flow velocity 'V' at the wire was evaluated using King's law

$$E^2 = A + BV^C \tag{3.5}$$

where E is the measured voltage, while A, B and C are the constants that are evaluated from hotwire calibration in a tunnel. A correction is made for changes in ambient temperature following Bearman (1971)

$$E_c = E_m\left(1 - \frac{\varepsilon}{(\Omega - 1)}\right)^{1/2} \tag{3.6}$$

After linearising the second term in the right hand side, the equation 3.6 is given as

$$E_c = E_m\left(1 - \frac{\varepsilon}{2(\Omega - 1)}\right) \tag{3.7}$$

where E_m is the measured voltage, E_c is the corrected voltage, Ω is the overheat ratio of the hotwire sensor and ε is the fractional change in air temperature from reference ambient conditions.

The technique of Kuroumaru *et. al* (1982) with the modifications suggested by Goto (1991) was used to establish the mean velocity vector at a given point. The technique relies on the calibration of the hotwire over a range of pitch and yaw angles. For the present research turbine, the hotwire probe was calibrated from -35^0 to 40^0 of pitch angle and between ± 120^0 of yaw angle. For each pitch angle a curve similar to the following form is fitted to the data:

$$\frac{V_{eff}}{V} = \left[I(\lambda)\cos\{Q(\lambda)\chi\} + R(\lambda)\sin\{Q(\lambda)\chi\}\right] \tag{3.8}$$

where V_{eff} is the effective velocity measured by the hot-wire and V is the actual velocity, I, Q and R are second order polynomial functions of the pitch angle λ (relative to the probe stem). The angle χ is measured between the velocity vector and the hotwire sensor normal. It is given as follows

$$\chi = \arcsin\{\sin\gamma.\cos\lambda.\cos(\zeta - \xi) - \sin\lambda.\cos\gamma\} \tag{3.9}$$

Figure 3.10 also defines the angles used in the calibration. In processing the experimental data, the calibration was applied to measurements of V_{eff} obtained at eight different probe positions. For the current experiments, these positions were 40^0, 60^0, 80^0 and 100^0 either sides of the estimated null position which is short prong of the hotwire facing the flow direction. The velocity vector is then obtained by using the method of least squares.

A ‘Dantec’ 55N20 signal conditioner was used to amplify the signal so that the full range of the analogue to digital converter could be utilised. The output of the conditioning unit was sampled at 50 kHz frequency to capture 128 points across each rotor pitch. Each ensemble was equivalent to two rotor blade pitches. The acquisition of each ensemble was triggered by the once-per-rev sensor and repeated 130 times to build up an ensemble average of the spatial non-uniformities in the rotor-relative frame. The accuracy of this technique relies on the rotor speed remaining constant. For sampling such a small sector of the rotor-relative flow this is a reasonable assumption but becomes less so as the overall sample duration increases. In the present investigation, the turbine speed was maintained $\pm$ 1.0 rpm of the design value. The slanted hotwire was also used behind the delta wing and the stator for measuring the unsteady flow field.

3.5.3 Three Axis Hot-wire

The development of the stator flow within the rotor blade passage was investigated in the relative frame of reference using a three-axis hotwire probe. The probe has a measurement volume of 2mm in diameter as shown in figure 3.11. The three axis hot-wire sensor is made up of three different single-axis inclined sensors arranged perpendicular to each other.

Due to the length-diameter ratio of the hotwire sensor (l/d = 200), it was not appropriate to use the ‘cosine law’ or its modifications to represent the response of the sensor at different angles of attack (Champagne *et. al.* (1967)). For this reason, a technique similar to that used in 5-hole pneumatic probe was developed. The technique relies on the interpolation of the data contained in a look-up table. A fixed, low turbulence, constant velocity jet was used for probe calibration in a calibration tunnel. The flow velocity at the wire was evaluated using King’s law with Bearman correction. The probe angular response was calibrated by varying the yaw and pitch angles of the probe (± 30 degrees) with respect to the calibration jet. The yaw and pitch angles as well as the three-sensor voltages and velocity magnitude were recorded at each angular position. Non-dimensional calibration coefficients were chosen to give good resolution over most of the calibrated range. The choice of representative pitch and yaw angle parameters depends on the orientation of the sensors relative to the probe co-ordinate system. A representative velocity parameter was found to be,

$$V_{rep} = \sqrt{\left(V_A{}^2 + V_B{}^2 + V_C{}^2\right)} \tag{3.10}$$

where, V_A, V_B, V_C are the apparent velocities measured by wires A, B and C respectively. The coefficients were defined as follows,

$K_y = (V_B{}^2 - V_C{}^2)/V_{rep}{}^2$ --- Yaw angle coefficient (3.11)

$K_p = (V_A{}^2)/V_{rep}{}^2$ --- Pitch angle coefficient (3.12)

$K_v = (V_{rep}{}^2)/V^2$ --- Total velocity coefficient (3.13)

The two non-dimensional coefficients K_P and K_Y were used as independent co-ordinates for the calibration. From the rig test data, the values of K_P and K_Y at a particular point in the area traverse were evaluated. Using the look-up table for these non-dimensional coefficients, the yaw angle, the pitch angle and the velocity of the flow were evaluated by linear interpolation. The sample calibration map as a function of yaw and pitch angle coefficients is given in figure 3.12.

3.5.4 Data Reduction

This section describes the definitions of statistical quantities used in high frequency measurements. All the measured voltages were converted to velocities before the determination of the statistical quantities. The acquisition of the data was triggered using a once-per-revolution signal. The ensemble-mean of N realisations of a quantity $\alpha(t, n)$ is then defined by

$$< \alpha(t) >= \frac{1}{N}\sum_{n=1}^{N} \alpha(t,n) \tag{3.14}$$

where time 't' is measured from a once-per-cycle datum point for a periodic process. The time mean of $\alpha(t, n)$ is denoted by $\overline{\alpha}$. The ensemble root-mean-square (rms) is defined as

$$rms = \sqrt{<< \alpha(t)^2 >>} = \sqrt{\frac{1}{N}\sum_{n=1}^{N}(\alpha(t,n) - < \alpha(t) >)^2} \tag{3.15}$$

It represents the amount of deviation, positive or negative, from the average value of the signal at that phase. The turbulence intensity data presented in this thesis represents the mean of all the three velocity components. This is given as

$$Tu_{rms} = \sqrt{\frac{1}{3}\left(<< V_x >>^2 + << V_r >>^2 + << V_\theta >>^2\right)} / V_{ref} \tag{3.16}$$

The time-mean rms value is determined using the equation,

$$\overline{Tu}_{rms} = \sqrt{\frac{1}{\tau}\sum_{n=1}^{\tau} Tu_{rms}^{2}} \qquad (3.17)$$

For the presentation of unsteady measurements, the time ‘t’ is non-dimensionalised by the stator wake passing period τ when measuring at rotor exit, and non-dimensionalised by the rotor wake passing period when measuring at stator exit.

3.6 Flow visualisation experiments

In the present investigation, Oil-and-dye flow visualisation was used for understanding stator blade flow patterns. Smoke flow visualisation was used in understanding the stator flow transport phenomena in rotor.

3.6.1 Oil and Dye flow visualisation

The surface streamline patterns give an insight into the structure of the flows within the turbomachinery blade passage, especially the secondary flows. Flow visualisation experiments were carried out using oil and dye for this purpose. During these experiments a thin coat of an appropriate mixture of fluorescent paint pigment and diesel oil was applied to the blades and surrounding endwall surfaces. During the experiment, the wall shear stress of the flow transports some of the applied mixture in the direction of the near surface-flow.

In each experiment, the rig was run for 45 minutes at the appropriate flow conditions. The resulting traces left by the flow on the investigated surfaces were examined and recorded by photographic means under ultra violet light for further analysis. The use of fluorescent pigments enables the use of very small amount of the paint to obtain a very clear picture. This makes the current technique less intrusive. Utmost care is essential in the application of the mixture, in the choice of oil with the right viscosity and in the duration of the experiment in order to minimise the effects of gravity and to maximise the responsiveness to local shear stress while still retaining enough of the mixture to be photographed.

3.7 Flow Measurements

The rig is equipped with a large amount of fixed instrumentation as shown in figure 3.13. The air inlet temperature is measured using a T-type thermocouple. The inlet total pressure is measured as the pneumatic mean of the pressures measured by the four Pitot tubes located at 200mm upstream of the stator leading edge. Two sets of 22 static pressure tappings located in the hub and casing at the same location as the pitot total pressure probes are used for the determination of the mass flow in the intake section. Downstream of the rotor there are two further sets of 12 static pressure tappings located at the hub and casing and equi-spaced around the annulus. Five static pressure tappings are also used on the rotor hub in the relative frame across one rotor blade pitch. The traverse ring at plane 1 is also equipped with one static pressure tapping. In addition to this fixed instrumentation, line and area traverses were carried out at planes 1 and 3 using variety of probes. These are described in this section. The uncertainty analysis associated with the measurement of pressure, loss coefficient and turbine efficiency is given in Appendix A1.

3.7.1 Inlet Boundary layer measurements

The inlet boundary layer was measured at a location 80% axial chord upstream of the stator leading edge for turbine 1 and turbine 2 and 80% stator axial chord upstream of the delta wing row for delta wing configuration. The boundary layer profile was measured with a flattened Pitot tube having a thickness of 0.15 mm. The probe was traversed radially from hub to casing in 178 points such that at least 30 points were measured inside the boundary layer thickness. The results for the turbine 2 are shown in figure 3.17 at hub ($\delta^*/h=0.008$, H=1.31) and casing ($\delta^*/h=0.0077$, H=1.30). The data is plotted in the form of a non-dimensional velocity profile. This boundary layer data is used as input for the numerical simulations. The blockage due to the hub and casing boundary layers were accounted while calculating the inlet mass flow rate.

3.7.2 Blade surface static pressure measurements

Surface static pressure tappings were used to measure the pressure distributions along the stator and rotor blade surfaces. Nylon tubing was cast into the surface of the rotor and the stator blades. The pressure tappings were created by cross-drilling into these pressure lines at 10, 25, 50, 75 and 90 percent span forming a matrix of pressure tappings on each blade surface. Fig 3.15 shows the pressure surface of the instrumented stator. Spanwise lines indicate the positions of the pressure lines. Each spanwise row of tubes was covered with a strip of 0.04 mm thickness adhesive tape which was wrapped around the leading and the trailing edges and covered the row of holes. To make measurements of the chord-wise static pressure distribution at a particular spanwise position, the adhesive tape was simply removed from the relevant row of holes ensuring that all others remained covered.

3.7.3 Area Traverses

Area traverses were carried out downstream of the blade rows and within the blade row using five-hole pneumatic, single slanted hotwire and three axis hotwire probes. In all of the measurements, the probes were small relative to the blades having diameters of less than 1.5 % of the blade pitch. The axes of the probes were aligned parallel to the mean flow direction in order to minimise the errors. Probes fitted to the traverse system were introduced into the turbine through the traverse ring. When a particular traverse ring was not being used, further precautions against leakage were taken by sealing the slot. Table 3.3 summarises the experimental traverses made during the course of the present study.

The probes were traversed radially from hub to casing and circumferentially over one blade pitch. Fine data grid resolution was used in regions of large total pressure gradients, such as in the blade wakes and secondary flows. The accuracy of the probe positioning is determined by the probe initial positioning, the linearity of the traverse mechanism and the stepper motor step resolution. Table 3.4 summarises the probe positioning accuracy for the present set of experiments. Important parameter, which characterises the flow losses in a blade row, is the coefficient of stagnation pressure loss. It is defined as

$$Y = \frac{P_{01} - P_{02}}{\frac{1}{2}\rho\left(\frac{V_{x1}}{\mathrm{Cos}74^0}\right)^2} \qquad (3.18)$$

Where P_{01} and P_{02} corresponds to the stagnation pressure at the inlet to the blade row and at the exit of the row respectively. The stagnation pressure loss is non-dimensionalised with a reference dynamic head. The velocity of the flow at the exit of the blade row is approximated by (V_{x1}/Cos74^0), where V_{x1} is the axial velocity at the inlet to the blade row and 74^0 was chosen as the representative exit flow angle. This definition of loss coefficient is used throughout the thesis unless mentioned otherwise. For the rotating frame measurements, the relative stagnation at the inlet to the row and at the outlet to the row is used for evaluating the loss coefficient. Since the rotor exit flow, as measured by a stationary probe, depends strongly on the relative position of the probe and the stator it was decided that a full unsteady area traverse of the rotor exit flow over one stator pitch was necessary. The area traverses were carried out while using the slant hotwire behind the rotor and while using the five-hole probe behind the rotating hub and behind the delta wing row.

3.8 Numerical Approach

The numerical simulations discussed in this thesis were performed with a steady Navier-Stokes solver 'MULTIP81' of Denton (1997) and an unsteady Navier-Stokes solver ‘UNSTREST’ also of Denton (1996). Both these numerical simulation methods are described in this section.

3.8.1 Steady Simulations with MULTIP

This code solves the three-dimensional modified Reynolds averaged Navier-Stokes equations on a structured, non-adaptive mesh. The equations of motion are discretised to second order accuracy and integrated forward in time. ‘MULTIP81’ uses the ‘SCREE’ scheme, which has been recently developed by Denton (1997). A mixing length model with wall function is used for modelling the turbulence in the flow. A full multi-grid method and local time stepping are used to accelerate the convergence. A mixing plane between blade rows allows the properties to mix out circumferentially at the downstream plane whilst preserving the spanwise variations.

A grid of 41x92x45 points for the stator and 41x99x45 points for the rotor was employed in the numerical simulations in the pitchwise, streamwise and spanwise directions respectively. The grid expansion ratio determines the rate at which the grid is stretched away from the solid boundaries until the maximum permitted cell size is reached. Grid expansion ratios of 1.3 near the endwalls and 1.2 near the blade surfaces were used in these computations. In order to represent the vorticity at the inlet to the stator blade row, a total of 9 cells were employed inside the endwall boundary layer thickness, which is of the order of 6% span.

All the numerical investigations were carried out at the design condition of the turbine (i.e. flow coefficient of 0.351). The turbine inlet axial velocity was 13.85 m/sec (i.e. M=0.038). Since this is very low for numerical calculations, it is increased by a factor of two. The non-dimensional design parameters were maintained by suitably changing the rotor speed and mass flow rate. The exit Mach number from the stator and rotor is approximately 0.28. The Reynolds number used in these simulations were the same as the experimental Reynolds numbers to correctly model the viscous effects on the blade surfaces and endwalls.

A large number of trials were carried out for different control parameters such as smoothing, time step, relaxation before an acceptable solution was obtained. In steady computations, convergence was deemed to have occurred when the average error reaches the specified limit and the ratio of local mass flow to inlet mass flow is close to one. Table 3.5 outlines the principle parameters used for the steady calculations performed during the present investigation. The mixing length factor is used in the evaluation of turbulent viscosity, which is the ratio between the maximum mixing length allowed, and the blade pitch.

3.8.2 Unsteady Numerical Simulations – UNSTREST

'UNSTREST' is an unsteady 3-D multi blade row code solving the Navier - Stokes equations in a simplified form. This is essentially an unsteady version of MULTIP with some important changes. The time stepping is performed using the Ni scheme (Ni (1982)). Multiple blade passages per row are calculated with the blade count adjusted to achieve an integer number of blades in each row. Each passage is calculated in turn and the resulting primary variables stored globally. This greatly reduces the storage requirements but prevents cusps from being easily used on leading and trailing edges. Once all the passages have been updated, periodicity is applied along the passage boundaries. A sliding interface plane between blade rows allows properties to be interpolated from one row boundary to another. The time step is uniform over the entire domain to achieve time accuracy and it is dictated by the stability limit imposed by the smallest cell in the domain. The pressure, density and velocity at the stator and the rotor exits were monitored to check for a periodic solution and to establish the number of time steps required in a blade-passing period.

During the later stages of the investigation, a modified version of 'UNSTREST' called 'UNSTSS9' was used. This solver uses the 'Scree' numerical scheme employed earlier in MULTIP81. Figures 3.16 and 3.17 show the computational mesh in the meridional plane and a blade-to-blade plane respectively. More grid points are placed near the leading and the trailing edges of the blade rows to resolve the flow properties more accurately. A detailed view of the grid at rotor mid-span near the leading edge is shown in figure 3.18. Table 3.5 outlines the principle parameters used for the steady and unsteady calculations performed during the present investigation. Table 3.6 summarises the various steady and unsteady simulations carried out in the present investigation for various test cases explained in section 3.3.

Vortex Transport in a Radially Stacked Turbine Stage

4

4.1 Introduction

This chapter presents a study of the vortex-blade interaction within the blade rows of a radially stacked high-pressure turbine. The steady flow field is investigated using the blade surface static pressure distributions and pneumatic probe measurements downstream of each blade row in both rotating and stationary frames of reference. Furthermore, unsteady measurements have been carried out using a three axis hot-wire. The transport mechanisms of the stator wake and passage vortices through the rotor blade row have been studied using smoke flow visualisation. Steady and unsteady numerical simulations have been performed using the structured three-dimensional Navier-Stokes solver to further understand the blade row interactions.

The steady flow field in the turbine is first discussed and the unsteady rotor flow field is discussed with the help of measurements and computations. The predicted flow field is investigated from the perspective of loss production. The contribution of the unsteady flow to the stage loss was evaluated using unsteady numerical simulations. The effect of stator viscous flow transport on the rotor flow angles is also discussed. Finally, a kinematic model is proposed for the transport of the secondary flow vortices through the downstream blade row based on the understanding obtained from the measurements and numerical simulations.

4.2 Steady flow field

The three-dimensional flow field is discussed with the help of measurements at planes 1, 2, 3 which were located at 20% C_x downstream of stator trailing edge, 60% C_x distance downstream of the rotor leading edge and 20% C_x downstream of rotor trailing edge, respectively. All measurements were undertaken at the turbine design flow coefficient.

4.2.1 Stator Flow

Area traverses at plane 1 were performed with a five-hole pneumatic probe. The stator exit flow is discussed using the contours of stagnation pressure loss (Y) (figure 4.1(a), see equation 3.18 for definition). The high loss region in the middle of the plot corresponds to the stator blade wake. Figure 4.1(a) shows that core flow away from the endwalls (60% of the span) is essentially two-dimensional flow. Most of the loss over the span is associated with the blade wake but there is also additional loss near the hub and casing due to the endwall secondary flows. These loss cores can be identified at 10% and 85% of the blade span situated on the suction side of the passage. At the casing, the loss core is more diffuse and the maximum pressure loss is 28% less than at the hub. The centres of the passage vortices are coincident with the maximum streamwise vorticity and these locations are indicated in figure 4.1(a) by the tips of the arrows. The centres of the passage vortices are located closer to the endwalls than the location of the maximum stagnation pressure loss. A local increase in loss can be seen at region 1 in figure 4.1(a). This is due to the interaction between the blade surface flows and the new end-wall surface flows at the suction surface corner.

At plane 1, the ensemble-mean velocities indicated that the maximum amplitude of the periodic velocity fluctuation is less than 1% of the mean value. This is because the potential field of the rotor is relatively weak. Contours of the time-mean phase-averaged turbulence intensity ($\overline{Tu_{rms}}$) derived from 3-axis hot-wire data are presented in figure 4.1(b). The mean turbulence intensity reaches a maximum value of 9% in the hub secondary flow, whereas it is 7% in the casing secondary flow. This indicates that the secondary flow at the hub is stronger than that at the casing, and is consistent with the five-hole probe measurements of figure 4.1(a). The centre of the wake has a relatively high turbulence intensity of 6%. In the free-stream region, the turbulence intensity is very small (around 0.7%). The time-mean values of the turbulence intensity (figure 4.1(b)) correlate well with the measurements of stagnation pressure loss (figure 4.1(a)). This indicates that the turbulence intensity is a good marker for identifying the flow structures.

4.2.2 Rotor Flow

Figure 4.2 presents the results of a relative frame five-hole probe traverse at plane 3. The data obtained from five radially disposed rotor leading edge Pitot tubes have been interpolated to provide a reference stagnation pressure for the traverse data at each radius. The high loss region near mid-span is the wake. Figure 4.2 shows that at least 40% of the span (from 30-70%) is occupied by essentially two-dimensional flow.

Near the hub, losses associated with secondary flows are evident. A large hub passage vortex can be seen covering up to 25% of the span. The loss associated with the tip leakage is centred at 95% span (figure 4.2). This loss core occupied almost 55% of the passage width and 10% of the span. This plot suggests that the tip leakage flow is the dominant secondary flow at this location.

Figure 4.3 presents the secondary velocity vectors from five-hole probe measurements at traverse plane 3. The secondary velocity vector is defined as the difference between the local velocity vector and the reference flow direction, which is 74^0 in this particular case. This angle is chosen because it coincides with the mean flow angle of the rotor hub and leakage vortices so that they become readily apparent. Various secondary flow features can be identified in this figure. At 90% and 20% blade span on the suction side of the passage (regions A & B), two clockwise rotating vortices are observed. These are the rotor hub passage vortex and the tip leakage vortex, respectively. In addition to these two vortices, near region D, a vortical structure rotating anti-clockwise can be observed. This structure could be either due to the interaction between the stator and rotor casing secondary flow or due to the rotor secondary flow. This is discussed further in section 4.4 using the unsteady measurements.

4.3 Smoke Flow Visualisation of Rotor-Stator Interaction

Smoke flow visualisation was carried out to identify the transport mechanisms of the stator wake and secondary flow features through the rotor blade row. The results from two particular experiments are discussed. Firstly, smoke was introduced through the stator trailing edge at blade mid-span to investigate the wake transport through the rotor passage. In the second experiment, smoke was introduced in the stator hub region to study the stator passage vortex transport through the rotor. Photographs were obtained using a stroboscope while holding the camera shutter open to obtain a total of eight superimposed exposures.

Results from the first experiment are shown in figure 4.4 in the form of a sequence of smoke flow visualisation pictures separated by equal intervals in time over one stator wake-passing period. The wake is identified by the white region in the middle of the passage. Figure 4.4(a) shows the wake just inside the blade row. The bowing of the wake is observed in figure 4.4(a). As the wake is drawn into the rotor passage, it is τconvected at the local free stream velocity. The bowing of the wake is due to the mean higher convection rate in the mid-passage flow compared to near the blade surfaces. By the time $t/\tau=0.286$ (figure 4.4(b)), the wake has been convected to 50% of the surface length on the suction side. The re-orientation or shearing of the wake can be seen in this figure. The shearing of the wake occurs because the fluid close to the suction surface convects at a much higher rate than the fluid near the pressure surface. This leads to the stretching on the pressure side leg of the wake (region 1). The wake centre line distortion and re-orientation continues at $t/\tau=0.571$ (figure 4.4(c)). By the time $t/\tau=0.857$ (figure 4.4(d)), the wake has been convected further towards the rotor trailing edge. The net result of all of the above phenomena is that the wake appears to be concentrated on the suction surface at the exit of the rotor passage with a 'tail' stretching back to the leading edge of the pressure surface of the adjacent blade (Hodson, 1985b, 1998). The same stator wake can be seen entering the adjoining rotor passage at time $t/\tau=0.857$ (region 2).

Results from the second experiment are shown in figure 4.5 as a sequence of smoke flow visualisation pictures at four equal intervals in time, over one stator wake-passing period. The pictures were taken at the exit of the rotor blade looking upstream and perpendicular to the throat as shown in figure 4.5(e). Each picture shows the flow across three blade pitches at any instant in time, designated as passages 1, 2 and 3, respectively. In these pictures, the stator passage vortex is again identified with the white regions.

Figure 4.5(a) shows the results at time $t/\tau=0.000$, the smoke appears on the suction surface inside the blade passage. The smoke structure is circular in nature and confined to the rotor hub region on the suction surface of the third blade passage. In the second blade passage, the smoke traces, which entered the blade passage at an earlier instant in time, can now be observed near the throat region. The structure of the smoke in this passage consists of two regions. One is circular in nature near the rotor hub and the other is elongated in the spanwise direction above the circular region. The smoke in the elongated region appears to be thinner. The smoke structure in the first blade passage has some similarity with the smoke structure in the second blade passage. The smoke traces from passage 1 appear downstream of the rotor trailing edge. The smoke pattern is thinner than in the previous passages indicating the dispersion of the smoke due to increased dissipation in the secondary flow.

Figure 4.5(b) occurs approximately after one-quarter of a stator passing period (t/τ=0.286). Since the previous time instant, the smoke trace in the third blade passage moved further downstream compared to figure 4.5(a). The smoke trace in the second passage convected further downstream to the rotor trailing edge. The radial extent of the smoke has increased. Smoke from the first blade passage moved downstream, appearing even more dispersed. Figures 4.5(c) and 4.5(d) show the progression of the incoming features of the stator secondary flow through the rotor blade passages, at subsequent time instants. At time t/τ=0.571 (figure 4.5(c)), the smoke trace at the exit of the first blade passage had almost disappeared, as the secondary flow of the stator move downstream. At t/τ=0.857 (figure 4.5(d)), smoke can be observed entering a new rotor passage designated as passage 4.

In general, it is observed that the secondary flow vortices are convected through the downstream blade row in a similar but not identical way to the wake. Due to the effects of vortex distortion, stretching and shearing, the vortically moving fluid appears to be concentrated on the suction side. A detailed description of the kinematic behaviour of the passage vortices is presented in section 4.5 after reviewing the results from unsteady measurements.

4.4 Stator Wake and Vortex Transport

The turbulence intensity, obtained from 3-axis hot-wire measurements, was used in tracking the stator flow inside the rotor. To better visualise the unsteadiness of the flow, the minimum value of the turbulence intensity in one stator wake-passing period at each traverse point has been subtracted from the local turbulence intensity. The result is given as the additional random unsteadiness

$$Tu_{aru} = \sqrt{Tu_{rms}^{\,2} - Tu_{min}^{\,2}} \qquad (4.1)$$

Figure 4.6(a)-(d) shows the contours of the additional random unsteadiness (Tu_{aru}) at plane 2 (rotor mid-chord), over one stator wake-passing period. Due to geometrical constraints, the traverse does not cover the full pitch. Figure 4.6(a) shows that the mid-span area of the suction surface is exposed to a low level of additional random unsteadiness. In figure 4.6, the minimum turbulence intensities were subtracted out from the total values. Therefore, if the rotor features are still observed in these plots, they represent unsteadiness in the rotor flow, due to the presence of the stator flow inside the rotor. The stator features can be observed to convect over one stator wake-passing period in these plots as well. The stator viscous flows such as blade wake and passage vortices can be identified with increased additional random unsteadiness. The remnants of a previous wake (region 2) can be seen near the pressure surface with higher values of additional random unsteadiness. Near the casing (region 1), the viscous flow originating from the casing in the stator row is beginning to appear at 85% span. At stator exit (figure 4.1(a)), the same secondary flow can be observed at 90% span and hence indicating the inward radial movement of the stator flow at this location.

Figure 4.6(b) occurs at one-quarter of a stator passing period after figure 4.6(a). Now the suction surface is exposed to the incoming wake (region 4). The minimum levels of turbulence intensity at this plane are less than 0.5% in the free stream region. This indicates that regions of low turbulence intensity separate the stator exit perturbations as they pass through the rotor blades. Apart from the wake and stator casing secondary flow, a feature that is identified with the stator secondary flow is also present in figure 4.6(b) (region 3). This is at a higher radius (20% blade span) than indicated by the stator exit traverse (10% blade span in figure 4.1(a)). Smoke flow visualisation shows that the upward radial migration of the flow is due to the effect of the stator secondary flow on the rotor flow near the hub. At $t/\tau=0.625$, the wake (region 4) now extends over the full span. The stator casing secondary flow is just beginning to disappear in the plot whereas the hub secondary flow is still present. The final plot (figure 4.6(d)) shows that the stator secondary flow now appears nearer to the pressure surface at a lower radius.

The differing convection rates on the pressure and suction surfaces means that the transport of fluid that lies near the suction surface is more rapid than that which lies close to the pressure surface. Since the traverse plane is located near rotor mid-chord, most of the distortion that this difference causes would have already taken place. Consequently, the stator wakes and secondary flow features are expected to appear first at this traverse plane, nearer the suction side of the passage. This is illustrated by the data presented in figure 4.6. The maximum turbulence intensity (Tu_{rms}) in the stator wake in plane 2 is observed as 3.8% (not presented in the thesis). This value is lower than the value at the stator exit (6.0%) indicating a decrease in turbulent kinetic energy of the stator wake between stator exit and rotor mid-chord. This implies that there is no turbulence generation due to vortex cutting in this turbine unlike the observation of Binder (1985).

Figure 4.7 presents the contours of the additional random unsteadiness (Tu_{aru}) equi-spaced in time, over one stator passing period, at traverse plane 3 (rotor exit). At time t/τ=0.125 (figure 4.7(a)), the additional random unsteadiness is low throughout the passage (0.5%) except in the rotor wake (region 1) and secondary flow (regions 2 and 3). At this time instant, the increased levels of additional random unsteadiness are beginning to appear near the casing (to the right of region 2) and the hub (outwards of region 3). Figure 4.7(b) occurs one-quarter of a stator passage period after figure 4.7(a). The unsteadiness has reached a maximum level of 6.5% to the right of the rotor trailing edge separated by a low turbulence region. Regions 4 and 5 are the stator casing and hub secondary flow vortices, respectively. Region 6 corresponds to the stator wake. The interaction between the stator flow and the rotor secondary flow is observed in figure 4.7(b) near the suction side of the blade, at around 25% and 70% of the blade span. It is also observed that region 5 is radially outward of region 3, indicating the radial movement of the stator flow features. By time t/τ=0.625 in figure 4.7(c), the stator flow features (region 8) appear right on the suction side with a very high additional random unsteadiness of more than 6.5%, at the hub. Near the casing, the rotor flow structure is more fragmented (region 7). This is due to the interaction of the rotor leakage and passage vortices with the stator wake and passage vortices. At time t/τ=0.875, the flow is slowly reverting to the values of t/τ=0.125 with the maximum additional turbulence levels of the stator features reducing (regions 9 and 10) and the free stream fluid returning to low additional turbulence values.

Figure 4.8 presents the contours of turbulence intensity ($\overline{Tu}_{rms}$) at rotor exit (plane 3) in one stator wake-passing period. The stator viscous flow features and their transport through the rotor blade row, observed in the figures 4.8(a)-(d), are similar to the figure 4.7(a)-(d). The stator flow is observed in figure 4.8(b) and the maximum interaction between the stator flow and rotor is observed at time t/τ=0.625. The stator flow is more clearly visible in figure 4.7 than in figure 4.8. In figures 4.6 and 4.7, the minimum turbulence intensities were subtracted out from the total values. Therefore, if the rotor features are still observed in these plots, they have to be due to the unsteadiness in the rotor wakes and secondary flows. This unsteadiness is due to the presence of the stator flow inside the rotor. These stator flow features influence the endwall boundary layers of the rotor and in turn influence the rotor secondary flow. By affecting the laminar-turbulent transition of the blade boundary layers, the rotor profile loss is also affected. By observing region 3 at all time instants in figure 4.7, it can be seen that the rotor secondary flow at the hub moves radially and circumferentially, varying in size over one stator-passing period.

The unsteady features of the flow are discussed with the help of hot-wire data (figure 4.9). Figure 4.9 shows the time varying secondary flow field with the help of secondary velocity vectors derived from hot-wire data. The secondary velocity vector is defined in the same way as in the steady case. At 20% and 90% blade span on the suction side of the passage (A & B), two clockwise rotating vortices can be seen at all time instants. These represent the rotor hub passage vortex and tip leakage vortex, respectively. At time $t/\tau=0.125$ and $t/\tau=0.375$, a counter-rotating vortex is observed just on the side of the rotor hub passage vortex (region C). This is due to the hub passage vortex of the stator. A close observation of the flow near the suction surface corner at time $t/\tau=0.625$ reveals the presence of a vortex (region D). This vortex is below the leakage vortex at 80% blade span, rotating in the opposite direction to the leakage vortex. As this vortex is not present at all time instants, it is probably due to the interaction between the rotor and stator secondary flow. The varying strength of tip leakage flow can be observed at all the time instants. This is due to the interaction of the suction side leg of the stator passage vortex near the casing with the leakage vortex, which is rotating in the same direction.

The contours of axial vorticity at plane 3 are shown in figure 4.10 at time $t/\tau=0.625$. The velocity gradients in the r and θ directions can only be calculated from the measured data at a given traverse plane. Hence, only the axial component of the vorticity is presented here. The axial vorticity is given by

$$\zeta_x = \frac{1}{r}\left(\frac{\partial(rV_\theta)}{\partial r} - \frac{\partial V_r}{\partial \theta}\right) \qquad (4.2)$$

The positive values of ζ_x indicate clockwise motion. The leakage vortex is confined to 15% of the span from the casing on the blade suction side. A region of negative vorticity can be observed just below the leakage vortex at 80% span. This corresponds to the interaction between the rotor and stator casing passage vortex. Another region of positive vorticity can also be observed below this secondary passage vortex at 75% blade span. This can be attributed to the stator passage vortex. At the hub, a region of positive vorticity can be observed near 22% span corresponding to the rotor hub passage vortex. The negative vorticity corresponding to the stator hub secondary flow can be observed above the rotor secondary flow region at 30% span.

4.5 A Simple Vortex Transport Model

A comprehensive picture of the wake-blade and the vortex-blade interactions can be assembled from the flow visualisation and unsteady measurements and simulations. The kinematic transport of the stator passage vortex through the rotor is discussed with the help of figure 4.11. The stator wake transport through the rotor blade row is also depicted at mid span.

The stator hub passage vortex is chopped by the downstream blade row in a similar way to the wake. It is then convected through the rotor passage at the freestream velocity. The pitchwise variation in convection velocity across the passage is responsible for the distortion of the vortex centre-line as shown in figure 4.11. The bowed vortex tube appears to have two counter-rotating legs extending back to the leading edges of the adjacent blades. These are termed as the suction side leg (region 3, figure 4.11) and pressure side leg (region 2, figure 4.11). Reorientation and shearing of the vortex occurs because a fluid particle will convect along the suction surface at a much higher rate than the pressure surface. As the suction side leg of the vortex is accelerated away from the pressure side leg, which remains in the vicinity of the leading edge, it stretches along the suction surface. The pressure side leg of the vortex stretches across the passage. It is expected that the stretching of the vortex tube will result in the reduction in the vortex diameter with concurrent increase in its angular velocity and subsequent increase in the dissipation rate. Due to the flow modifications (distortion, stretching and shearing), the stator hub vortex appears to be concentrated on the suction side with a tail extending up to the rotor leading edge on the pressure side of the passage.

The presence of the rotor passage vortices affects the transport of the stator passage vortices through the rotor. The kinematic interaction between the stator and the rotor passage vortices has two effects. Firstly, the suction side leg of the stator passage vortex is displaced radially upwards over the developing rotor passage vortex near the hub. The pressure side leg of the stator passage vortex is counter-rotating to the rotor hub passage vortex. Additionally, the pressure side leg of the stator passage vortex, rotating in the same direction as the rotor passage vortex, is engulfed by the rotor passage vortex. Similar phenomena are observed at the rotor tip, where the tip leakage and passage vortex interaction causes the stator features to move towards the mid-span on the suction surface.

4.6 Comparison of Steady and Unsteady Simulations

Steady and unsteady Navier-Stokes simulations of the turbine stage were carried out using the solvers described in chapter 3. Identical grids, numerical schemes, mixing length parameters, relaxation parameters and boundary conditions were used for the steady and unsteady numerical simulations. In unsteady simulations, the upstream viscous features pass through the downstream blade row. In order to reduce the computation time and data storage, the unsteady simulations were carried out with 42 stator blades and 42 rotor blades instead of 36 and 42 blades of stator and rotor respectively. For a better comparison, the steady calculations were also carried out with 42 stator and rotor blades. This small variation in stator solidity has only a small effect on the rotor flow field as illustrated in the later part of this section.

Figure 4.12 shows the stagnation pressure loss coefficient (Y) contours at plane 3 derived from steady numerical simulations, unsteady numerical simulations and five-hole probe measurements. The relative stagnation pressure values at the rotor leading edge were averaged in the pitchwise direction. These values were used in evaluating the relative stagnation pressure loss at rotor exit, assuming cylindrical stream surfaces for the steady and unsteady computations. The general flow structure agrees well with the measurements both qualitatively and quantitatively. The leakage vortex (region 1) is dominant at this location in both the steady and unsteady simulations and in the measured data. The steady simulations over-predict the leakage losses, while the unsteady simulations compare well to the measured loss. The difference is due to the interaction between the rotor leakage flow with the periodically varying stator flow field, which is modelled, only in the unsteady simulations. The rotor blade wake is wider in both the steady and unsteady computations compared to the measured data. This may be due to the difference in the rate of mixing in the wake in both the computations and experiments, which arises from the mixing model. The loss core corresponding to the secondary flow at rotor hub (region 2) can be observed at 25% blade span. The flow structure appearing at 80% blade span (region 3) is due to the stator-rotor interaction. Steady simulations did not predict the strength and location of this feature, while the unsteady simulations predicted this structure well. Overall, there is good agreement between the measurements and the unsteady simulations. This increases the confidence in the unsteady numerical simulations and the conclusions that can be drawn from them.

The use of pitchwise averaging in a three-dimensional flow such as this turbine destroys much of the flow detail. Any radial displacement of the flow feature will significantly change the pitchwise averaged profile. However, since a downstream blade row will usually be designed to accept the pitchwise averaged flow from an upstream blade row, it is instructive to compare the results in this way, in order that any limitations may be noted.

Figure 4.13 presents the spanwise distributions of the pitchwise averaged rotor exit relative yaw angle, the envelope of its variation from the mean and the stagnation pressure loss coefficient. Results from the measurements, and both steady and unsteady numerical simulations are compared in this figure. The familiar features of the secondary flow with overturning near the end-wall and underturning towards the mid-span can be seen in the relative yaw angle at the hub (figure 4.13(a)). Near the casing, the flow is underturned due to the leakage vortex. A small underturning followed by overturning can be observed at 80% span. This is due to the stator-rotor interaction as explained in figure 4.9. Accordingly, the steady simulation does not predict this flow feature. At blade heights corresponding to the strongest stator secondary flow, differences between the steady and the unsteady computations are greatest (5^0 difference). The maximum difference between the steady and unsteady predictions of absolute flow angle is of the order of 10^0 at rotor exit at 25% and 80%

blade span. This illustrates the limitations of steady flow calculations. The time average unsteady hot-wire data are in good agreement with the steady five-hole probe data except near the tip region. This is because the hotwire measurement accuracy is compromised in this region, due to the relatively large gradients in yaw angle and the large random unsteadiness. Furthermore, the unsteady computations are in good agreement with the measured data. This also shows that the small variation in stator solidity in numerical simulations has only a small effect on the flow field at rotor exit. Figure 4.13(a) also shows that there is no underturning at the endwalls and overturning towards the mid-span for this particular turbine (unlike that of Sharma *et. al.* (1988)).

Figure 4.13(b) shows the envelope of the phase-averaged variations around the time mean value of relative yaw angle. The levels of unsteadiness over most of the blade span is in agreement with the measured unsteady data except near the hub. The amplitude of the fluctuation is much less than the difference between steady and unsteady computations indicating that the non-linear effects are due to the vortex transport rather than simple periodic fluctuations. Figure 4.13(c) shows the comparison of stagnation pressure loss coefficient between measurements and steady and unsteady computations. In calculating the relative stagnation pressure loss coefficient, it is assumed that the stream surfaces are cylindrical. The local increase in the loss can be observed corresponding to the hub and casing passage vortices regions and in the tip leakage regions. There is a reasonable agreement between the five-hole probe measurements and the time averaged unsteady computations. The steady computations are not able to predict the losses accurately near the rotor secondary flow regions and tip leakage flow.

4.7 Unsteady Loss

The predicted flow field was interrogated from the perspective of loss production to determine the contribution of the unsteady flow to the time-average performance of the stage. The only accurate measure of loss in an unsteady flow is provided by the increase in entropy (Denton (1993)). All the entropy produced within the flow field will eventually pass through the exit boundary of the stage and be perceived as the stage loss. Various definitions are possible in defining unsteady loss. If the steady flow field is defined as the flow with steady inlet to the turbine stator row and turbine rotor row, it is not possible to measure the steady loss or unsteady loss experimentally. This is because the rotor exit flow field is a result of the interaction between the rotor and upstream stator at all times during one stator-passing period. It is easier to evaluate the steady loss in numerical simulations for the designer by using a mixing plane approach. The unsteady loss in the present investigation is defined as the difference between the average entropy flux passing through the exit boundary, in one wake passing cycle, of an unsteady calculation and the corresponding entropy flux from a mixing plane steady calculation. This is given as,

$$\Delta\omega_u = \frac{1}{\tau}\left\{\int_0^\tau\int_0^p\int_0^r 2\pi r(\rho V_x s)\,dr\,dy\,dt\right\}_u - \left\{\int_0^p\int_0^r 2\pi r(\rho V_x s)\,dr\,dy\right\}_s \tag{4.3}$$

where 's' is the specific entropy of the fluid and 'τ' is the time for one stator passing period. The difference between the integrated entropy fluxes from steady and unsteady computations were used in evaluating the contribution of unsteady loss to the stage efficiency. This is given as

$$\Delta\eta = \left(\frac{\dot{m}\,\Delta h_0}{\dot{m}\,\Delta h_0 + T_0\,\omega}\right)_s - \left(\frac{\dot{m}\,\Delta h_0}{\dot{m}\,\Delta h_0 + T_0\,\omega}\right)_u \tag{4.4}$$

The efficiency measured for the turbine stage under investigation is 88.3% with an uncertainty of $\pm 0.8\%$. Further details of the uncertainty analysis are given in Appendix A1. The stage efficiencies calculated from steady and unsteady computations were 89.8% and 86.6% respectively. The contribution of the unsteady flow to the stage loss is about 3.2 percentage points of efficiency, which is about 1/3rd of the steady loss. The accuracy of calculated stage efficiencies depends on various control parameters like the numerical scheme, the computational grid, the grid resolution in resolving various viscous phenomena, smoothing factors, relaxation factors and mixing length parameters. In the present investigation, all the factors effecting the stage efficiency are the same in both steady and unsteady simulations. Although the absolute magnitudes of calculated values may differ from measurement values, the relative difference between the two types of simulations can be trusted. The measured (88.3%) and predicted (86.6%) efficiency values for the present turbine agree reasonably well, giving confidence to the predicted values of the unsteady loss.

Although the unsteady loss seems relatively high, it should be noted that the additional loss includes contributions from other steady phenomena. For example, it is known that when the wake passes through the downstream blade row, the flow incidence to the blade row changes temporally. Additional loss will be generated due to this effect, though it is likely to be small in this high-pressure turbine. In the mixing plane steady calculations, the wakes and passage vortices are instantaneously mixed out at the exit boundary of the blade row. Conversely, in unsteady simulations, these viscous structures are transported through the downstream blade row, generating additional losses due to wake mixing and vortex stretching. In the turbine under investigation, unsteady measurements discussed in section 4.4 confirmed the considerable effect of the stator flow on rotor blades. Hence, the main contributions to the unsteady loss is due to wake mixing, vortex stretching and the subsequent effect of these phenomena have on the development of rotor secondary flow in the downstream blade row.

4.8 Conclusions

The development of steady and unsteady three-dimensional flow in the turbine 1 has been described. The transport of viscous flow features within the rotor blade row has been analysed with smoke flow visualisation and three axis hotwire measurements. It is observed that the stator secondary flow vortices are convected through the downstream rotor blade row in a similar but not identical way to the wake. At the hub, the kinematic interaction between the stator and the rotor passage vortices has two effects. Firstly, the suction side leg of the stator passage vortex is displaced radially upwards over the developing rotor passage vortex at the hub. Additionally, the pressure side leg of the stator passage vortex is entrained into the rotor passage vortex. A simple kinematic model is proposed for the transport of the secondary flow vortices in the downstream blade row based on the understanding obtained from the measurements and the numerical simulations.

The time averaged unsteady measurements are in good agreement with the steady measurements except near the tip region. Unsteady numerical simulations were found to be successful in predicting the flow accurately near the secondary flow interaction regions. Comparisons between the steady and the unsteady numerical simulations with measurements highlighted the need for unsteady computations. The contribution of the unsteady flow to the stage loss has been evaluated using unsteady numerical simulations.

The Three Dimensional Flow Field in a High Pressure Steam Turbine Stage with Compound lean

5

5.1 Introduction

A considerable portion of efficiency debit in a low aspect ratio high-pressure turbine is due to the secondary flow (Harrison (1989)). Several researchers have investigated the mechanism of the suppression of secondary flows by three-dimensional stacking and blade sweep both experimentally and analytically (Potts (1987), Hourmouziadis and Hubner (1985), Walker (1987), Harrison (1989)). This chapter describes the steady and unsteady flow field of a leaned and swept high-pressure steam turbine. In the turbine under investigation, Mitsubishi Heavy Industries of Japan carried out the aerodynamic design of the blading. The blade design is aimed at producing a geometrical arrangement that can reduce the secondary flow losses. This chapter also highlights the sources of unsteady loss generation and provides an insight into the mechanisms of the interaction between the stator and rotor of a modern high-pressure turbine.

The steady flow field is investigated using the blade surface static pressure measurements, miniature pneumatic probes downstream of each blade row and oil-dye flow visualisation on the blade surfaces. The unsteady flow field of the rotor was investigated using a single-slant hot wire probe and a three-axis hot-wire probe. Steady and unsteady numerical simulations were performed using the structured 3D Navier-Stokes solver to further understand the flow field. Unless otherwise indicated, all the experiments and computations have been carried out at the design operating condition. The data from the numerical simulations was also used to identify the sources of loss generation. Concluding remarks are made in the last section of this chapter.

5.2 Blade Loading and Overall Performance

The turbine stage loading and overall performance are evaluated using blade static pressure measurements on the stator and the rotor blade surfaces and torque measurements at various flow coefficients.

5.2.1 Blade Loading

A series of experiments were performed to quantify the loading distribution along the blade at 10%, 25%, 50%, 75% and 90% span. The surface static pressures have been non-dimensionalised to give a static pressure coefficient (C_p). This is given as

$$C_p = \frac{P_{0in} - P}{\frac{1}{2}\rho U_m{}^2} \tag{5.1}$$

where P_{0in} is the inlet stagnation pressure and U_m is the rotor blade speed at mid span. Figure 5.1 presents the static pressure coefficient distribution on the stator blade surface plotted as a percentage of surface distance at various blade heights. The symbols represent the data from measurements and the solid line from numerical simulations. The maximum value of C_p at mid-span is 2.03 while the values of C_p at 10% span is 1.825 and 1.585 at 90% span. The locations of the peak suction at 10% blade span is observed at around 70% surface distance, 28% surface distance at 50% span and 58% surface distance at 90% span. These values indicate that the stator blade is aft-loaded near the ends and front-loaded in the middle of the span. The stator blade is off-loaded at the endwalls to reduce the secondary flow.

There is a very good agreement between the computations and the measurements indicating that the blade loading levels were represented correctly in the calculations. The difference between the suction and pressure side static pressure coefficient values near the stator leading edge at 75% and 90% span indicated the positive incidence angles of the flow near these regions. This is further illustrated with the help of flow visualisation and velocity vector plots discussed in section 5.3. These figures show that the flow stagnates on the pressure side of the blade surface, then accelerates around the leading edge to the suction side.

Figure 5.2 presents the static pressure coefficient distribution on the rotor blade surface plotted as a percentage of the surface distance at various blade heights. The results indicate that the rotor blade is also off-loaded near the endwalls ($C_{p, max}$ of 1.64 at 10% blade span and 1.64 at 90% blade span) compared to the mid-span region ($C_{p, max}$=1.784 at 50% blade span) to reduce the rotor secondary flow. Also the rotor blade ends are aft-loaded (the peak suction is at 52% and 44% of surface distance at 10% and 90% blade span respectively) and the mid-span sections are front-loaded (peak suction is at 37% of surface distance). A small amount of flow diffusion is observed at

90% span near the leading edge of the rotor blade indicating a positive at this spanwise location. The good overall agreement between the computations and measurements at all heights of the rotor and stator blade gave confidence in the calculations.

5.2.2 Overall Performance

The overall performance of the turbine was evaluated at different flow coefficients. The stage inlet conditions are determined using the free-stream values at stator inlet. The exit conditions are determined by radially traversing a 3-hole cobra probe far downstream of the rotor so that the flow is essentially axisymmetric. In these experiments, the radial traverses at the rotor exit were performed at a traverse plane located 186mm downstream of the rotor which is approximately 2.2 times the rotor axial chord. The exit static pressure was measured by the rings of pressure tappings fitted at the hub and the casing at two axial chords downstream of the rotor.

The definitions given in table 5.1 were used for evaluating the stage performance of the turbine. The measurement of efficiency represents a particular challenge in low speed facilities. This is due to the accuracy of the torque measurements. The tare of the torque meter was first set to zero at zero torque. A check on the accuracy of this setting was then provided by carrying out a test that is similar to the Willan’s test for the reciprocating internal combustion engines. The LSRT was operated at constant flow coefficient but at varying mass flow rates. If the effects of Reynolds number can be ignored, then the efficiency should remain constant. Under these circumstances, a plot of the measured torque against $\left(P_{01} - \overline{P_{exit}}\right)$ should be linear and pass through the origin. Experiments showed that by extrapolating to zero pressure difference the tare agrees with the zero torque value to within 0.1% of the torque at the design point of the LSRT.

The test described above provides a check on the tare of the torque meter. This is important for ensuring the repeatability of the measurements. Unfortunately, it does not provide a measure of the windage torque. To determine the windage torque, the following procedure of Lewis (1993) was adopted. This procedure relies on number of assumptions. Firstly, the efficiency must be constant (or nearly constant) over a suitable range of flow coefficients that include the design point. The initial estimates of the efficiency revealed that this is the case. Secondly, the windage torque must not be a function of the mass flow rate. Thirdly, the density at each measurement location must remain constant as the flow coefficient changes. Under these circumstances, the quantity

$$\frac{\rho_{in}\Omega T}{\dot{m}\left(P_{01}-\overline{P_{0exit}}\right)}=\frac{\rho_{inlet}\Omega\left(T_{measured}+T_{windage}\right)}{\dot{m}\left(P_{01}-\overline{P_{0exit}}\right)}=\eta \tag{5.2}$$

has a constant value. The true torque is given by the symbol T in equation 5.2.

An experiment was conducted in which the mass flow rate was varied but the rotational speed, and therefore the windage torque, was held constant. The above equation can be rewritten as

$$\frac{\dot{m}}{\rho_{in}}\left(P_{01}-\overline{P_{0exit}}\right)=\frac{1}{\eta}\Omega T=\frac{1}{\eta}\Omega\left(T_{measured}+T_{windage}\right) \tag{5.3}$$

The left-hand side of the above equation represents the isentropic power output. The right hand side of the equation represents the actual power output. The slope of the equation 5.3 gives $1/\eta$ of the turbine. The windage torque may then be determined by plotting the left-hand side of the above relationship against the quantity $\Omega T_{measured}$ and determining the intercept with the ordinate when the abscissa is zero. This is shown in figure 5.3 for the present turbine. A linear regression analysis provided the straight line fit to the data. It should be noted that at this point the term windage torque includes the torque generated by the rubbing seals, as it is virtually impossible to separate the two effects in the present configuration. In this present test turbine, the windage torque is calculated as 51.82 N-m.

Figure 5.4 presents the overall performance of the stage such as stage pressure coefficient, stage loading coefficient and total-total efficiency as a function of the flow coefficient. The efficiency measured for the turbine stage under investigation is 91.92% with an uncertainty of ± 0.8 percentage. Further details of the uncertainty analysis corresponding to the stage efficiency measurement are given in Appendix A1. The stage efficiency is almost constant with varying flow coefficient whereas the stage loading and stage pressure coefficient increased linearly with the increase in flow coefficient.

5.3 Stator Flow Visualisation

The use of oil and dye flow visualisation on blade and endwall surfaces gives valuable insight into the structure of the flow within the blade passage. Further details of the visualisation technique are outlined in section 3.6. Figure 5.5(a) show the flow visualisation on the suction surface of the stator. The line of sight has been chosen to be nearly perpendicular to the blade chord line and is slightly tilted towards the casing. In this way, the full extent of the blade as well as the blade to casing interaction can be observed. Red dye was applied on the blade suction and pressure surfaces and green dye on the endwalls.

It is observed from figure 5.5(a) that the suction surface flow is not two-dimensional and significant radial flows exist near the hub and the casing. The hub and casing endwall flow (green dye) can be observed on the blade suction surface. The radial migration of the endwall flow is more deeper near the casing than near the hub. The blade C_p distributions discussed earlier show that the maximum static pressure coefficient near the hub is higher than near the casing indicating higher blade loading near the hub compared to near the casing. The lower blade loading near the casing combined with larger radial migration of the endwall flow indicates that the secondary flow near the casing is weaker than near the hub. Two hub passage vortex lift-off lines (A and B) were observed on the suction surface. The lift-off line ‘A’ is due to the main passage vortex while the second line ‘B’ corresponds to the turning of the new endwall boundary layer developed after the lift-off of the main passage vortex.

An interesting feature observed in the flow visualisation is the radial jump (figure 5.5(a), region 1) on the suction surface. The radial jump extends the influence of the endwall flow to nearly 40% of the blade span from the casing. The axial location of this radial jump coincides with the blade throat. Hence, the affected area corresponds to the region uncovered by the preceding blade. In this region, the streamwise wall shear stresses are reduced due to the back surface diffusion. This reduced shear stress combined with the existing radial pressure gradient is thought to enhance the radial migration of the secondary flow on the suction surface.

Figure 5.5(b) shows the flow visualisation on the pressure surface of the stator and on the casing endwall. The flow on the pressure surface is two-dimensional with an overall tendency to move downward (from casing to hub). The lift-off line corresponding to the main passage vortex on the casing endwall can be observed from the pressure side of the blade to reach the suction side of the next blade around mid chord position.

Figure 5.6 present the flow visualisation on the hub endwall. Two stator blades have been removed from the hub in order to make the hub endwall more accessible. The lift-off line around the blade leading edge, associated with the horseshoe vortex, is clearly visible. At hub, the lift-off line reaches the suction surface from pressure side of the blade by 1/3rd of the axial stator chord. This is much earlier than the casing endwall, which indicates higher blade loading at the hub.

The flow from the pressure surface (region 2 in figure 5.5(a)) can be observed on the suction surface. This is due to the high positive incidence of the flow near the tip section resulting in a small re-circulation region. This effect was also observed in the static pressure coefficient distribution in figure 5.1 (90% span). The numerical simulations also predict the flow movement from PS to SS as shown in figure 5.7. Figure 5.7 presents the absolute velocity vectors of the flow near the stator leading edge at 90% blade span. It can be observed that the stator inlet flow stagnates on the pressure surface and accelerates around the leading edge to the blade suction surface. The extent of this region is about 20% of the blade span near the casing while the streamwise extent is very small. Hence, the effect of this region to the pressure distribution is restricted to the vicinity of the leading edge.

5.4 Stator Exit Flow

The flow field at the stator exit is discussed with the help of measurements obtained at a plane located at a distance 13% of C_x downstream of the stator trailing edge. Area traverses have been performed with a five-hole probe and a three-axis hotwire. The measurement techniques and the probe details are outlined in section 3.5. Figures 5.8 – 5.10 show the contour plots of the stagnation pressure loss (Y, see equation 3.18 for definition), secondary velocity vectors and absolute yaw angle compared with computed results (steady flows assumed) over two stator pitches. The stator blade wake can be identified with the high loss region in the middle of the figure 5.8(a). Most of the loss over the span is associated with the blade wake but there is also additional loss near the hub and the casing. The accumulation of the high loss fluid in these regions is a consequence of the hub and casing secondary flows. These loss cores can be identified at 10% and 90% of the blade span situated on the suction side of the blade passage. There is a good repeatability of the loss contours throughout the measurement domain of two stator passages. Figure 5.8(b) presents the loss coefficient predictions from steady numerical simulations. A comparison between the measured and computational results reveals a good qualitative and quantitative agreement with the main features of the stator flow being well captured. There is some discrepancy in the location and magnitude of the loss in the centre of the wake at 20% and 70% of blade span but the qualitative agreement is, nevertheless good. The computed width of the wake is more than the measured wake width. This may be due to the difference in the rate of mixing in the wake in the computations and in the experiments, which arises from the mixing model.

The measured and the computed secondary velocity vectors at the stator exit plane are shown in figure 5.9. The secondary flow velocities have been calculated using the following equation

$$\vec{V}_{secondary} = \vec{V}_{local} - \vec{V}_{local}.\hat{e}_{mean} \tag{5.4}$$

where $\vec{V}_{local}$ is the velocity determined from the 5-hole probe data at each point and $\hat{e}_{mean}$ is the unit vector in the mean exit flow direction. The secondary flow vectors can also be defined using blade exit angle or based on inviscid flow direction at various spanwise locations. All the definitions have given similar picture. At the measurement location, the mean exit flow direction is α_{2mean}=74.51^0 and β_{2mean}=1.87^0, where α and β is the absolute yaw and pitch angles of the flow. In figure 5.9, two counter rotating vortices are observed on the suction side of the blade at 10% and 90% of the blade span. These are the stator hub passage vortex and casing passage vortex respectively. It can also be observed that there is a good agreement between the computations and measurements.

The absolute yaw angle contours of the flow field at stator exit plane both from measurements and computations are presented in figure 5.10. The mean flow direction at this plane is 74.5^0. With respect to this angle, the flow near the hub and casing endwalls is overturned as shown in figure 5.10. The underturning of the flow can be observed above the underturned flow regions corresponding to the secondary passage vortices. In the wake region, a distinctive pattern exists. On the blade suction side, the flow is overturned, and on the pressure side, the flow is underturned. The majority of the flow field is well predicted except near the wake regions. This again may due to the type of mixing model used in the present calculation. The good agreement between the experimental and the calculated flow angles indicates that the global rotation of the bulk flow is well captured by the calculation.

The use of pitchwise averaging in a three-dimensional flow such as in this turbine looses much of the flow detail. Any radial displacement of a flow feature will significantly change the pitchwise-averaged value. However, since a downstream blade row will usually be designed to accept the pitchwise averaged flow from an upstream blade row, it is instructive to compare the results in this way, in order that any limitations may be noted. Fig 5.11 presents the radial distributions of the pitchwise averages of the absolute yaw angle and stagnation pressure loss coefficient at plane 1 behind the stator.

The first plot (figure 5.11(a)) shows the absolute yaw angle distributions in the spanwise direction. It can be observed that there is a small variation of yaw angle from 30 % to 65 % of the span. However, near to the hub and the casing the familiar features of the passage vortex with overturning at the endwalls and under-turning towards the mid-span are observed. The spread of the secondary flow in the radial direction is more than that would occur with a conventionally designed blade. The second plot (figure 5.11(b)) shows the variation of the stagnation pressure loss coefficient in the spanwise direction. The loss coefficient is high at hub and casing due to the endwall boundary layers. The local increase in loss coefficient is observed corresponding to the secondary vortex cores near the hub and casing. There is a very good agreement between the measurements and the computations as can be seen in figure 5.11(b).

One of the advantages of the accurate numerical computations is that it enables us to investigate the flow inside the blade row, which is otherwise impossible to measure in the experiments. One particular example of the advantage of computations is explained below. Figure 5.12(a) shows the details of the secondary velocity vectors, defined with respect to the computational grid direction, near the casing region at a distance of 91% of stator axial chord downstream of the blade leading edge. It can be observed that near the pressure surface and casing endwall corner, there is a small re-circulating region. Investigation of static pressure contours at the same location (figure 5.12(b)) revealed that this was due to the radial static pressure gradient existing in the blade due to the blade lean. The re-circulation region corresponds to a higher static pressure than the static pressure at the lower spanwise location. This increases the radial migration of the flow near the casing towards the blade midspan and hence develops a re-circulation region. This shows that the blade lean at the tip and hub section may have to be reduced to produce the optimum performance from this blade in the contest of using three-dimensional concepts.

5.5 Rotor Exit Flow

The flow field at the rotor exit is discussed with the help of measurements at plane 3, which is located at a distance of 26% rotor C_x downstream of the rotor trailing edge in the relative frame of reference. Area traverses at plane 3 have been performed using a 5-hole probe and three-axis hot-wire. The measurement techniques and the probe details are outlined in section 3.5.

The loss coefficient (Y, see equation 3.18 for definition) contours of the relative stagnation pressure are presented in figure 5.13 and compared with the computed results. The data obtained from five radially disposed rotor leading edge Pitot tubes have been interpolated to provide a reference stagnation pressure for the traverse data at each radius. The high loss region near the mid-span is identified with the rotor blade wake. Figure 5.13(a) shows that at least 45% of the blade span (from 20% - 65%) is occupied by two-dimensional flow but there is additional loss near the hub and casing. The accumulation of the high loss fluid in these regions is a consequence of the hub and casing secondary flows and due to shroud leakage. At the hub, in addition to the loss core corresponding to the hub secondary flow on suction side of the blade, there is another loss core located near the pressure side of the blade (region 1). This is discussed further in section 5.6 with the help of the unsteady measurements. At this measurement location, the shroud leakage loss dominates 15% of the blade height from casing. The casing secondary flow is pushed radially inwards by the growing leakage flow.

Figure 5.13(b) presents the contours of the loss coefficient predicted with a steady CFD calculation. The predicted hub and casing secondary flows are stronger than the measured values and are radially moved outward near the hub and inward near the casing. The loss core corresponding to the shroud leakage is not to be seen in this plot, as this feature is not modelled in this computation. Initially the shroud leakage modelling was not thought to be important, as there was believed to be very little chance of leakage from rotor leading edge to rotor trailing edge. This is because there are five radial seals and two axial seals to discourage the leakage flow. The comparison of experimental data and computations revealed that this leakage flow is important and has to be modelled. The geometry of the rotor shroud and its arrangement is discussed in section 3.4.

Numerical simulations were carried out by including the modelling of shroud leakage flow (Denton (1999)) and the associated loss. In this shroud leakage model, the leakage flow is estimated from the seal clearance and number of seals and is bled off from the main flow. The change of angular momentum of the leakage flow due to friction on the shroud and casing are estimated using input values of the skin friction coefficient. The work done on the shroud by the leakage flow is also calculated. The leakage flow is then injected into the main flow downstream of the blade row and the conservation equations determine the mixing loss. Figure 5.13(c) presents the results of this computation at the same measurement location. The loss core due to shroud leakage flow can now be observed very near the casing above the secondary flow loss as observed in the experiments. The computed magnitude of the shroud leakage loss is 20% more than the measured value. Given the simplicity of the model used for shroud leakage the comparison between the computations and experiments is thought to be very good especially in terms of the location and magnitude of this loss core corresponding to shroud leakage flow. The efficiencies of the turbine stage with and without including the shroud leakage flow are predicted from steady simulations. They are given as 94.13% without shroud modelling and 92.74% with shroud modelling. In comparison, the measured efficiency for the turbine stage is 91.92% with an uncertainty of $\pm 0.8\%$. A good agreement between the measured efficiency and the predicted efficiency with shroud modelling can be observed. The difference in efficiencies between the two simulations with and without shroud leakage modelling is 1.4 %, indicating the importance of modelling the leakage flow in simulations.

It was noted above that the predicted location of the loss core due to the hub secondary flow was found to be at a higher radius than measured. A close examination of the secondary air path of the rig indicated that there is a possibility of flow leakage from mainstream to the nose bullet (the lowest pressure region in the test rig) at the gap between the rotating and the stationary parts of the hub drum. A rubber V-seal was used to prevent this leakage flow. Although it was not possible to measure the amount of leakage, an attempt has been made to understand the effect of hub bleed on secondary flow development.

The latest version of the ‘MULTIP’ called ‘MULTALL’ (Denton (2000)) has the provision for accounting the bleed flows. Simulations were carried with this code and the results are presented in figure 5.13(d). The effect of hub bleed on the development of the hub secondary flow can be clearly observed with the location of the hub secondary flow at much lower radii. Further refinement of the prediction can be carried out by varying the amount of bleed flow (0.9% of total mass flow in the present case). This small exercise explains the importance of secondary flows like shroud leakage and bleed on the overall performance prediction. As the unsteady version of ‘MULTIP’ does not include shroud leakage and bleed flow models, results from the numerical simulations not incorporating these models are used for comparing the steady and unsteady flow fields.

Figure 5.14 presents the secondary velocity vectors of the flow at the rotor exit plane from measurements and computations. The secondary flow velocities have been calculated using the equation 5.4. Various secondary flow features can be identified in this figure. At 10% and 90% blade span on the suction side of the passage, two counter rotating vortices are observed. These are the rotor hub passage vortex and casing passage vortex respectively. The discrepancy in the radial location of the passage vortices at the hub and the casing can be observed between the steady computations and measurements. A vortical flow rotating in opposite to the main passage vortex (region 1) can be identified on the pressure side of the blade. This location also corresponds to the loss core on the pressure side, which was discussed in the figure 5.13(a). This is discussed further in the following section with the help of the unsteady measurements.

5.6 Unsteady Flow in the Rotor

It is evident from the measurements of the flow field at stator exit that the rotor will experience fluctuations in the inlet flow. This unsteadiness arises due to the relative motion of the rotor with the stator. The spatial gradients in the blade-to-blade direction at the stator exit are seen as temporal disturbances by the rotor row as the rotating blades sweep past the stator blade row. It is useful to characterise the potential importance of unsteadiness by evaluating the reduced frequency parameter (for definition, see equation 2.2). It represents the number of wakes (or other stator features) found in a single rotor passage at any instant of time. The reduced frequency for the present turbine is 1.67 indicating that the unsteady effects can be significant in the turbine stage.

The work presented in this section was undertaken to give an insight into the mechanisms of the interaction between the stator exit flow and the rotor. Extensive time resolved data has been obtained downstream of the rotor (26% C_x downstream of the rotor TE, plane 3) using a single-slant hot wire probe (SSHW) in the absolute frame of reference. The method used for the data acquisition and data reduction is outlined in section 3.5.

5.6.1 Single Slant Hot-wire Data

An area traverse was performed in the stationary frame with a SSHW over one stator pitch from 2.1% to 97.1% of the blade span. The results are presented below. The measurement grid consisted of 21 equi-spaced points in the pitchwise direction, 28 points in the spanwise direction and 300 points over two rotor pitches. This gave rise to a three dimensional data (r, θ, t) set that can be viewed in various cutting planes.

Figure 5.15 presents the rotor exit absolute velocity contours in constant radius plane at blade mid-span. The time variation of the ensemble averaged velocity trace at a point in the measurement grid was expressed as rotor pitch and plotted as abscissa. The circumferential distance was plotted as a fraction of stator pitch. The rotor wake can be identified as a thin region of low absolute velocity surrounded by the higher velocity fluid as can be observed in figure 5.15(a). The variation if any, of the velocity in the rotor wake in stator pitch direction can be attributed to the influence of the stator blade. One such region is observed between 75% to 100% of the stator pitch, associated with low velocity covering a full rotor pitch. At this stator pitch region, the rotor wake can not be identified separately. A region of velocity deficit (75-100% stator pitch) with respect to local absolute velocity can be observed in the middle of the rotor blade passage corresponding to the stator flow. A quantitative assessment of this stator influence on the rotor flow is given in the following section.

Figure 5.15(b) shows the variation of the absolute velocity of the flow with rotor pitch (across section AA) at 25% of the stator pitch. The narrow regions of low absolute velocity are due to the rotor wake. The mass average value of the absolute velocity at this pitchwise location is 15.9 m/sec. By comparing the average velocity at various circumferential locations, the effect of stator influence on the rotor flow can be evaluated. Figure 5.15(c) shows the variation of the absolute velocity with stator pitch at section BB. The large velocity deficit corresponding to the stator flow interaction can be observed towards the rear end of the stator pitch as described earlier. The velocity in this region is 12m/sec i.e. around 4m/sec less than the average velocity at section AA. The large magnitude of the velocity deficit at 75-100% stator pitch indicates the significance of the unsteady interaction of the stator flow with the rotor.

The measurements carried out at six radial locations in the blade span are presented in figure 5.16. The probe is traversed radially and the measurement grid is same at all the radial locations. Hence, the rotor passages start at the same tangential location (abscissa) at each radius. This figure also shows the absolute velocity contours with time expressed as rotor pitch and stator pitch. The low velocity region can be observed between 30-55% stator pitch at 10% span compared to 75-100% pitch at 50% span and 10-60% pitch at 76.5% span indicating the three dimensionality of the stator flow influence on rotor blade. It can also be observed at 35.2 % span and 76.5% span, the stator flow influence covers almost full rotor blade pitch and the rotor blade wake can not be distinguished with the rest of the flow. This may be due to the interaction of the stator passage vortex with the downstream rotor flow. This interaction is further investigated by analysing the data in the measurement plane in both absolute and relative frames of reference.

The measurements carried out at various radial and circumferential locations in the area traverse were assembled together in the absolute frame and presented over one rotor blade passing period in figure 5.17(a). Figure 5.17(a) presents the secondary flow lines in a quasi-orthogonal plane. The secondary flow lines were obtained by allowing the imaginary fluid particles to convect in the rotor exit secondary flow field for short period of time. The lines represent the response of the fluid particles to the secondary flow field at the traverse plane and are not a depiction of the flow development within the blade passage. At time $t/\tau = 0.0$, two rotor passage vortices corresponding to the hub and casing can be observed at locations 1 and 2. After $t/\tau = 0.4$, the same vortices are moved towards the left side of the figure and the new set of passage vortices from the adjoining rotor blade can be observed at locations 3 and 4. The extent and the strength of the passage vortices 1 and 2 are altered as the rotor sweeps through the stator field. At $t/\tau = 0.8$, the passage vortices 1 and 2 are no longer visible in the measurement window and the vortices 3 and 4 move towards the centre of the measurement window.

The unsteadiness in the rotor flow field is visualised better by analysing the data in the relative frame of reference as shown in figure 5.17(b). The time trace of each velocity signal is expressed as rotor pitch and plotted at all the spanwise locations over one stator pitch. The influence of the relative stator location on the rotor flow field can be understood from these plots. Figure 5.17(b) shows the secondary flow lines in the relative frame of reference. In these plots the stator flow features will be appearing and disappearing periodically at the frequency of the stator blade passing. At $t/\tau = 0.0$ (where τ is the stator blade passing period), the regions 1 and 2 correspond to the rotor passage vortices at hub and casing. At time $t/\tau = 0.2$, the region 3 which is rotating opposite in direction to the rotor passage vortex can be observed on the pressure side of the blade. The same region 3 can be observed at a higher radius of 40% span when $t/\tau = 0.0$ and at 24% span when $t/\tau = 0.6$ and at 18% span when $t/\tau = 0.8$. Further analysis of the absolute velocity and turbulence intensity data at these spanwise locations corresponding to region 3 reveal that at time $t/\tau = 0.2$, this rotating structure is strongest and the rotor passage vortex (region 1) is much smaller. At time $t/\tau = 0.6$, the rotating structure (region 3) is smaller and the rotor hub passage vortex (region 1) is much larger. The sign of the rotation near the region 3 and its periodic nature indicate that this region may be due to the hub passage vortex of the previous stator.

Figure 5.18 presents the contours of the turbulence intensity in relative frame of reference over one stator blade passing period. At time $t/\tau = 0.0$, the regions 1 and 2 corresponding to the rotor hub and the casing passage vortices can be observed. The region 3 corresponding to the interaction of the stator flow with rotor as discussed in the figure 5.17(b) can also be observed. The region 3 is at a much higher radius near

40% span at $t/\tau = 0.0$ and moves down to around 20% span at $t/\tau = 0.8$. This region 3 becomes stronger at time $t/\tau = 0.2$, which also corresponds to the lower magnitude of the hub passage vortex of the rotor as shown in figure 5.17(b). This indicates that the presence of the stator secondary flow at rotor exit reduced the secondary flow corresponding to the rotor. The region 3 further discussed using the flow predictions from the unsteady numerical simulations in the following section.

5.7 Comparison of Unsteady Measurements with Unsteady CFD Predictions

The results of the three-dimensional time-accurate calculations (UNSTREST) have been used to give insight into the unsteady interactions occurring within the blade row. Additional details of the calculation and the grid employed are given in section 3.8. Owing to a very fine grid in the endwall regions and near the blade surfaces, large number of iterations (200,000) are needed before obtaining a periodic solution. The pressure, the density and the velocity at a grid point in the flow field are monitored to check for a periodic solution and number of time steps required in a wake-passing period. The same grid as that used for steady simulations was employed for unsteady simulations. All the numerical investigations were carried out at the design condition of the turbine (i.e. flow coefficient of 0.351).

The time-mean solution is obtained by averaging all the flow properties in one wake passing period (5188 iterations in this case). The time mean rotor exit flow field is discussed with the help of the figure 5.19(a) and 5.19(b). Figure 5.19(a) presents the contours of the relative stagnation pressure loss coefficient at the exit of the rotor, at the same axial plane as in figure 5.13. The difference between the mixing plane steady calculations and time mean calculations can be observed in the location of the rotor secondary flow at the hub and the casing. The loss core near the hub is at a much lower radii than in the steady calculation. Hence, the previous assumption of not modelling the hub bleed in the calculation being the reason for the difference in location of the secondary flow is incorrect. This unsteady calculation shows that the interaction between the stator flow and rotor flow is responsible for radial location of the hub secondary flow.

Figure 5.19(b) compares the measured pitchwise averaged relative flow angle with the predictions from steady simulations and time mean unsteady simulations. The overturning near the hub and underturning towards the midspan is due to the hub secondary flow. The location of the hub passage vortex is predicted from unsteady simulations to be near 10% blade span, which compares well with the measured value of 13%. The magnitude of the yaw angle variation from time mean data also agrees well with the measured angle variation in the hub region. The corresponding location for the hub passage vortex from the steady computations is observed at 20% blade span. Initially it was thought that the flow leakage at the hub may be responsible for the discrepancy between the predicted and measured location of the hub secondary flow but the unsteady predictions indicate otherwise. It was also established using the unsteady measurements that there is a strong interaction between the stator secondary flow and the rotor flow at hub. All these observations indicate that the rotor flow field is better predicted by the time accurate calculation than the mixing plane steady calculation.

Figure 5.19(b) also show that there is a large difference between the measurements and predictions of flow angle near the casing. This difference is due to the shroud leakage flow, which is not modelled either by steady computations or by the time accurate computations. The interaction between the stator flow and rotor flow near the hub is further investigated using the instantaneous flow field at five equi-spaced time steps from unsteady simulations in figure 5.20 over one stator wake-passing period.

The flow field is described using the entropy and streamwise vorticity contours. Any irreversible flow process creates entropy and so inevitably reduces the isentropic efficiency. Entropy is also a good marker of the flow. The entropy rise through a blade row can be written as

$$\Delta s = R \ln\left[\frac{P_{01r}}{P_{02r}}\left(\frac{T_{02r}}{T_{01r}}\right)^{\frac{\gamma}{\gamma-1}}\right] \tag{5.5}$$

where R is the universal gas constant and γ is the ratio of specific heats. For an adiabatic flow, and provided there is no change in radius between the states 1 and 2 in a rotating system, the relationship between the entropy change and relative stagnation pressure ratio simplifies to $P_{02r}/P_{01r} = e^{(-\Delta s/R)}$. This function $e^{(-\Delta s/R)}$ is referred to as the entropy function and calculated from the unsteady numerical simulations based on the instantaneous flow. Vorticity is another good marker of the flow. The calculated velocity gradients in the r, θ and z direction can be used to calculate the three components of the vorticity. The radial, axial and tangential components of the vorticity can be combined to calculate the streamwise vorticity (ζ_{sw}) in the mean flow direction.

Figure 5.20(a) presents the contours of the entropy function and figure 5.20(b) presents the contours of the streamwise vorticity at four time instants in one wake passing period. The entropy function value of 1.0 represents the no loss region and lower values represent the loss corresponding to the wake and secondary flow regions in the flow. The positive values of ζ_{sw} indicate the clockwise rotation of the fluid and the negative values indicate the anti-clockwise rotation of the fluid.

Various secondary flow features can be observed at time $t/\tau = 0.0$ in regions 1, 2 and 3. Regions 1 and 2 correspond to the rotor hub and casing secondary flow and can be identified in streamwise vorticity plot as two counter rotating flow regions. The region 3 can be observed to the right of region 1 and is associated with a vortical structure, rotating opposite to region 1 as seen in vorticity plots. The corresponding entropy function plot also shows that the region 3 corresponds to a loss core indicating that this region may be due to the hub secondary flow of the previous stator. After one-quarter of a wake passing period at time $t/\tau = 0.25$, the region 3 is observed to decrease and then disappear completely by time $t/\tau = 0.50$. It can also be observed at time $t/\tau = 0.50$, region 1 covers the full rotor pitch indicating the amplification of rotor hub secondary flow and corresponding loss contours strengthen this argument. After another one-quarter of the wake passing period at time $t/\tau = 0.75$, the region 3 appears again to the right side of region 1. Region 3 pushes the rotor secondary flow towards the suction surface and in turn reducing the strength of the rotor passage vortex at hub. These observations are in agreement with the measurements described in figure 5.17 and 5.18. The varying strengths of the casing secondary flow corresponding to region 2 can be observed indicating the interaction of the stator casing secondary flow with the rotor flow field.

A comparison of the predicted pitchwise averaged unsteady flow field is made with the measurements from the five-hole probe traverse at the exit of the rotor (plane 3) in figure 5.21. The unsteady rotor flow field in one stator wake-passing period is presented at five instants in time separated by equal intervals. Figure 5.21(a) presents the spanwise distributions of the outlet to inlet ratio of the axial velocity density. This ratio is a representation of the outlet to inlet mass flow distribution in the spanwise direction. The mass flow passing through the hub and casing passage vortices regions is higher due to flow underturning in these regions. The change in mass flow distribution is negligible from 20% span to the blade tip at all instants in time. There is a considerable change in the mass flow corresponding to the hub passage vortex region between the various time instants in one wake passing period. This variation is due to the interaction between the stator flow with the rotor, while this interaction is minimal at all the other spanwise locations.

The relative yaw angle distributions at the exit of the rotor are presented in figure 5.21(b). Near to the hub and the casing, the familiar features of the passage vortex with overturning at the endwalls and under-turning towards the mid-span are observed. A very small variation in the yaw angles is observed between the various time instants from 70% span to the blade tip. The variation between the time instants is observed from 25% span to 70% span but there is negligible difference in the corresponding mass flow distributions in figure 5.21(a). This indicates that the difference in yaw angle is due to the variation in the lift of the rotor blade as the stator wake convects through the rotor blade. There is a considerable variation in flow angle between the time instants near the hub region. This reinforces the previous conclusion that the effect of the interaction of the stator passage vortex at the hub is significant in the turbine than near the casing. The flow visualisation, the static pressure distributions and the measurements at the stator exit indicate that the secondary flow at the casing in the stator blade is lower than the secondary flow at the hub region. The numerical simulations indicate that the reduced secondary flow at the casing reduced the unsteady variations of the mass flow and the corresponding flow angles while interacting with the rotor blade row. The circles indicate the results from the five-hole pneumatic probe measurements at the same plane. A good comparison between the measurements and the unsteady numerical simulations can be observed upto 70% span from the hub. The difference between the measurements and the simulations near the casing is due to not modelling the shroud leakage flow in the simulations.

Finally, the spanwise variation of coefficient of relative stagnation pressure loss is presented in figure 5.21(c). The predicted relative stagnation pressure at the rotor inlet in the middle of the passage at various radial locations provided the reference stagnation pressure in evaluating the loss coefficient. The measured relative stagnation pressure using the rotor leading edge Pitot tubes at five radial locations provided the reference pressure in evaluating the loss coefficient from the measurements. A large magnitude of the loss is predicted very near the hub due to the boundary layer on the hub endwall. The local increase in loss coefficient near the hub and casing secondary flow regions can be observed. A large difference in loss coefficient between the various time instants from 25% span to 70% span is observed. Similarly, the difference between the five time instants near the hub region is due to the effect of the stator vortex transport through the rotor row. The local increase in the loss can be observed at 75% blade span corresponding to the passage vortex. The difference between the predictions and the measurement from 73% span to the blade tip arises from not modelling the shroud leakage flow.

A good agreement between the measurements and the numerical simulations can be observed with respect to the interaction between the stator flow with the rotor (region 3). The predictions indicated the effect of unsteady flow on the mass flow distribution, exit yaw angles and the loss coefficient. Further investigation of the sources and locations of the loss generation is carried out section 5.8.

5.8 Loss Audit in the Turbine Stage

The historic breakdown of loss into ‘profile loss’, ‘endwall loss’ and ‘leakage loss’ continues to be widely used although it is now clearly recognised that the loss generation mechanisms are seldom independent. The relative magnitudes of the above three categories of loss are dependent on the type of turbine and other details such as blade aspect ratio and tip clearance. Generally, these losses are expressed in loss coefficients. The most common definition is the stagnation pressure loss coefficient (Y). The main disadvantage of using this definition is that when used in a rotating blade row, the relative stagnation pressure can change as a result of change in radius without there being any implied loss of efficiency. Denton (1993) clearly illustrates that the only accurate measure of loss in a flow is entropy. Entropy is a particularly convenient measure because, unlike stagnation pressure, stagnation enthalpy or the kinetic energy, its value does not depend upon the frame of reference.

By evaluating the entropy creation or generation in a control volume and summing many such control volumes in a blade passage, it is possible to calculate the entropy increase for the whole blade row. It can be shown from the energy equation (Hughes and Gaylord (1964)) that for a volume of 'vol', with a surface area 'A', the rate of entropy production due to viscous dissipation can be written as,

$$\int_{vol} \sigma \, dvol + \int_{A} \frac{k \nabla T}{T} \vec{n} \, dA = \frac{\partial}{\partial t} \int_{vol} \rho s \, dvol + \int_{A} \rho s \vec{V} \vec{n} \, dA \tag{5.6}$$

where σ is the entropy production rate per unit volume due to viscous shear and k is the thermal conductivity. The unit normal vector $\vec{n}$ is positive when directed out of the volume and the velocity vector is denoted as $\vec{V}$. As the present investigation is in a low-speed research turbine involving only incompressible and adiabatic flows, the entropy generation due to heat transfer is negligible. With these assumptions, equation 5.6 becomes

$$\int_{vol} \sigma \, dvol = \frac{\partial}{\partial t} \int_{vol} \rho s \, dvol + \int_{A} \rho s \vec{V} \vec{n} \, dA \tag{5.7}$$

Since the numerical predictions were found to agree well with the experiments (see figures 5.1-5.2, 5.8-5.11 and 5.13-5.14), the entropy production rate for the rotor and the stator blade rows was calculated from simulations. The intention was to identify the sources and regions of loss generation inside the blade row.

Figure 5.22(a) shows the specific entropy contours in a blade to blade plane at the stator mid-span location indicating the losses in blade boundary layers and in the blade wake. A typical control volume with the inlet and outlet entropy fluxes is depicted in figure 5.22(a). After evaluating the entropy fluxes into and out of each cell (control volume) of the computational domain, equation 5.7 was used to calculate the entropy production rate in that cell. Using the method of Jennings and Shin (1993), the whole blade was divided into the number of regions as shown in figure 5.22(b). The 10% of the span from the endwalls of the blade are defined as the hub and the casing region such that it will contain most of the endwall boundary layers. The pitchwise extent of the 'pressure surface' and 'suction surface' was taken as 10% of the blade pitch so that it will contain most of the blade boundary layer. The spanwise extent of these regions starts from the top of hub endwall region to the bottom of the casing endwall region. Two separate regions of upstream and downstream are also defined to account for the whole of upstream region from inlet to the blade leading edge and the downstream region from blade trailing edge to the mixing interface plane. The rate of entropy generated inside the whole blade row was taken as 100% and the entropy production rate in various regions of the blade was calculated as a percentage of the blade value.

Figure 5.22(b) shows the rate of entropy generated in various regions of the flow in the stator and the rotor though the total values of the entropy generated in the stator and the rotor are different. The total loss generated in the stator is 0.056 (J/Sec. K) and in the rotor is 0.076 (J/Sec. K). The maximum percentage of the rate of entropy is generated in the suction surface of the blade (44.3% for the stator, 31.4% for the rotor) compared to the other regions of the blade. The entropy generation rate in the casing region of the rotor (32.4% for the rotor, 19.6% for the stator) is much higher than the stator indicating a very strong rotor secondary flow at casing. The entropy generation rate in the ‘downstream’ region corresponds to the wake and vortices mixing loss. The large magnitude of the rotor mixing loss (36.7% for the stator, 52% for the rotor) is due to the larger downstream area considered in the analysis for the rotor. It is also due to the mixing of the deeper rotor wake and the passage vortices in the downstream of the rotor. This method of identifying the loss generating regions can be a useful tool for the blade designer as it enables him to decide the area of the blade row to be focussed on while optimising the new blade design.

A similar analysis was carried out using unsteady numerical simulations, where the entropy generation rate in each control volume was integrated over one stator passing period. Figure 5.23 presents the integrated entropy generation rate per unit volume ($\dot{S}'''$) results of such analysis in the rotor blade row. The magnitude of the entropy generation rate is being very large, logarithmic of the entropy generation rate per unit volume is presented in Figure 5.23(a) at the rotor mid-span radius in the blade-blade plane. The entropy can be seen produced in the surface boundary layers of the rotor as well as during wake mixing with the free stream flow. In addition to these, the entropy generation rate can also observed in the middle of the passage. Further investigation of the entropy generation rates was carried out at section A-A in a quasi-orthogonal plane. Figure 5.23(b) presents the logarithmic of the entropy generation rate per unit volume contours at section A-A. The entropy generation can be observed on the rotor suction surface due to the surface boundary layers. The regions corresponding to the rotor hub and casing passage vortices can also be observed. In addition to the above-mentioned regions, a region in the middle of the passage is also observed. This is the region, which corresponded to the loss observed in figure 5.23(a) and is near the rotor blade throat. This unsteady loss-generating region may be due to the compression of the stator wake near the rotor throat region, while convecting in the rotor blade row. It is described as follows.

Figure 5.24(a) presents the instantaneous entropy function contours at the rotor mid-span radius in the blade-blade plane. The stator wake can be identified in the middle of the rotor passage. The stretching of the stator wake near the rotor pressure surface and the shearing of the stator wake near the suction surface can be observed in this figure. Consider a segment BB of the stator wake at the rotor inlet, while convecting through the rotor blade from the conservation of mass flow inside the wake, it can be shown that it modifies to $B'B'$ near the rotor throat region. This results in the compression of the wake in the middle of the blade passage. If the wake is considered of as being two vortex sheets, from Kelvin's theorem it can be shown that the velocity difference is amplified if the wake is compressed. This increased velocity difference inside the wake in turn increases the mixing losses and hence the generation of unsteady loss as predicted in figure 5.23(b). Figure 5.24(b) presents the instantaneous entropy function contours across section AA of figure 5.24(a). In addition to the rotor wake, passage vortices, a region corresponding to the interaction between the stator flow (stator blade wake and passage vortices) and rotor can be observed in the middle of the passage. The two other entropy-generating regions observed in the middle of the blade passage in figure 5.23(b) can be identified to the stator hub and casing passage vortices from this figure. This unsteady loss generation is from the stretching of the stator passage vortices in the rotor blade and the loss generation can be observed at all the locations throughout the rotor blade. It is not possible to evaluate only the unsteady loss, as specifying the unsteady loss regions at each axial location is difficult. Instead, similar analysis of figure 5.22(b) is carried out by defining the regions corresponding to the suction, pressure surfaces and the hub and the casing and integrating the entropy generation rate in those regions. Figure 5.24(c) presents the results of the analysis. The entropy generation rate is 67.5% of the whole blade for the suction surface region. This is higher in comparison to 31.4% for the same region from the steady simulations. The entropy generation rate in the core region is 8.8% of the total entropy generated in the rotor blade row. The entropy generation rate in this region combined with some of the increase in suction surface region can be assumed to be due to the unsteady loss. This analysis has demonstrated the further use of the unsteady numerical simulations in identifying and understanding the mechanisms of unsteady loss generation in the turbine stage.

5.9 Steady and Unsteady Flow field in 'Rotating Hub' test configuration

The turbine tested in this configuration was same as the one described in the earlier sections except that this turbine has a rotating hub attached before the stator. This is done to simulate a rotating endwall before the stator at hub. Steady measurements were performed with a five-hole probe and the unsteady measurements with a three-axis hotwire. Area traverses were performed in the stationary frame over one stator pitch behind the stator and in the rotating frame over one rotor pitch behind the rotor. The details of the measurement grid and the blade span measured are given in table 3.3. The stator exit and rotor exit flow field is discussed with the help loss and turbulence measurements at the respective measurement planes.

5.9.1 Stator Exit Flow

The figure 5.25(a) presents the contours of the stagnation pressure loss coefficient behind the stator (8.4% C_x downstream of stator trailing edge, plane 1). This measurement plane is different to the datum configuration, which is at 13% C_x downstream of the stator trailing edge. The measurement has to be carried out in this plane due to the geometric constraints. Figure 5.25(b) shows the time mean turbulence intensity contours measured at the same location with a three axis hot-wire probe. The stator secondary flow is much stronger than the datum configuration (figure 5.8(a)) with a larger loss core near the hub. This is due to the skewed inlet boundary layer coming from the rotating hub to the stationary stator row. The loss structure is not much different compared to the datum configuration from 30% span to casing. The time-mean turbulence intensity contours in figure 5.25(b) correlate well the contours of the stagnation pressure loss. This again indicates that the turbulence intensity is a good marker for identifying the flow structures. The turbulence intensity in the middle of the stator wake is around 7% while in the middle of secondary flow regions are

10%. The integrated loss coefficient behind the stator for the datum case is 0.0210 (13% C_x chord downstream of stator trailing edge), while the loss coefficient for the rotating hub is 0.0246. The uncertainty associated with the measurement of loss coefficient in the stationary frame of measurements is given as ± 0.0015. Further details of the uncertainty analysis are given in Appendix A1. The difference in loss coefficient between the datum and the rotating hub configuration can be attributed to the increase in loss due to the skewed incoming boundary layer. This is calculated as 17.1 (± 7.1) % of the datum loss. The closer measurement plane to the stator trailing edge may have also contributed to the increase in the loss coefficient. The results from the present experiment show that the inlet skew has a significant effect on the development and migration of the passage vortex and loss core. This is similar to the findings of Bindon (1980), Boletis *et al.* (1983) and Walsh and Gregory-Smith (1987).

5.9.2 Rotor Exit Flow

The flow field at the rotor exit is discussed with the help of measurements at a new plane 3 which is located 10% C_x downstream of the rotor trailing edge in relative frame of reference. This measurement plane is different to the datum configuration, which is at 26% C_x downstream of the stator trailing edge. A five-hole probe with an extension length of 105mm was used to measure the rotor exit flow field very near to the rotor trailing edge. Figure 5.26(a) presents the contours of the relative stagnation pressure loss coefficient at plane 3. The rotor wake can be identified with the high loss region in the middle of the blade passage. It can also be observed that the rotor wake is two dimensional from 30-70% blade span and the width of the wake is much thinner than in the datum configuration. This is due to the difference in the location of the measurement planes between the datum and rotating hub configurations. The loss cores corresponding to the secondary flows at the hub and the casing are observed. The loss core corresponding to the leakage flow can also be identified. Another loss core is observed on the pressure side of the blade at 35% span (region 1) similar to the datum configuration. The origins of this region are discussed further in the section with the help of the unsteady measurements.

Figure 5.26(b) shows the phase averaged time mean turbulence intensity contours (Tu) derived from three-axis hotwire data at the exit of rotor (plane 3). The measurements are carried out in a relative frame of reference. The turbulence intensity in this figure and in all the subsequent figures in this thesis is defined by the following equation

$$Tu = \sqrt{\frac{1}{N}\sum_{n=1}^{N}\left(V_x(t,n) - V_x(t)\right)^2 + \left(V_r(t,n) - V_r(t)\right)^2 + \left(V_\theta(t,n) - V_\theta(t)\right)^2} \Big/ V(t) \qquad (5.8)$$

where 'n' represent the ensemble number in a total 'N' ensembles and time 't' is measured from a once-per-cycle datum point for a periodic process. The velocity V(t) is the ensemble mean of the instantaneous velocity at a point in the measurement grid.

The time mean turbulence intensity reaches a maximum of 18% in the middle of the shroud leakage flow whereas it is 17% in the hub secondary flow. The turbulence intensity in the centre of the wake is 13% and can be identified in the middle of the plot. In the free stream region, the turbulence intensity is around 7%. There is a large core of high turbulence intensity seen on the pressure side of the blade at 40% span (region 1) with a magnitude of 14%. This region covers full rotor pitch from the pressure side of the blade to the top of the hub passage vortex on the suction side of the blade. Further analysis of this region is presented below in relation to figure 5.28. The region below the shroud leakage flow is due to the rotor casing secondary flow and has 13% turbulence intensity. The mean turbulence intensity contours present a similar picture to the relative stagnation pressure loss data (figure 5.26(a)) again confirming the use of turbulence intensity as a marker in the flow.

The contours of ensemble averaged time-mean relative velocity at the same rotor exit plane 3 are presented in figure 5.27(a). The rotor wake, the hub and casing secondary flow and shroud leakage flow can be identified with low velocity regions and are marked in the figure. Figure 5.27(b) presents the relative yaw angle of the flow at the same location. The regions of the hub and the casing secondary flow are associated with flow overturning near the end walls and flow underturning towards the mid-span. The yaw angle in the region 4 is associated with the underturning at the bottom and overturning towards the mid-span in contrast to the classical secondary passage vortex indicating that this vortical structure is rotating in opposite direction.

Figure 5.28 presents the instantaneous turbulence intensity (Tu) contours plotted over one stator wake-passing period. The rotor wake and secondary flow features can be identified with the regions of high turbulence intensity (regions 1, 2 & 3 respectively). It can be observed that region 4 which is on the pressure side of the blade at 40% span and is varying with period equal to that of stator passing. The maximum value of turbulence in the region 4 can be observed at $t/\tau = 0.0$ and it gradually reduces to a minimum value at $t/\tau = 0.50$. By observing the region 2 at all the time instants, it can be said that the rotor secondary flow at hub moves radially and circumferentially, varying in size in one stator blade-passing period. The loss measurements with a five-hole probe indicate that this region 4 correspond to a relatively higher loss. The yaw angles and secondary velocity vectors show that this region corresponds to a vortical flow rotating opposite in direction to the main passage vortex. Furthermore, spectral analysis of the velocity trace in the centre of this region 4 showed a very strong stator blade passing frequency (330Hz) content indicating that this region is associated with the passage vortex of the stator at the hub.

The location of this turbulence core on the pressure side is at a much higher radius than on the suction side. This is slightly different to the vortex transport model proposed in chapter 4. The stator passage vortex tube, when it convects to the downstream rotor row, develops two counter rotating legs of the vortex on the pressure and suction surface of the rotor passage. Both legs of the vortex convect downstream with the local flow velocities. At the exit of the rotor in the present investigation, the pressure leg of the stator passage vortex was radially displaced upwards and the suction leg was entrained into the rotor passage vortex. This was due to two different factors. The first factor stems from the three-dimensionality of the blade and the existing static pressure gradient, which drives the flow near the pressure surface towards the mid-span. The larger radial migration of the vortex on the pressure surface than on the suction surface can also be explained with the help of vortex dynamics.

The vortical flows in the presence of solid surface are often analysed with the help of *method of images.* Wherein two or more flows in an unbounded fluid are arranged so that a streamline is formed coinciding with the position and the shape of a desired solid surface. The streamline is then replaced with a solid surface to give the desired solution. The flow structures behind the solid surface are referred to as images. Their only purpose is to create a streamline that can be defined as a solid surface. The incoming stator vortex rotating in the anti-clockwise direction, near the walls generates image vortex rotating opposite in direction inside the blade wall. The two vortices are of equal strength but rotating in opposite directions form a vortex pair as shown in figure 5.28(a). Each induces a velocity of same magnitude and direction in the other and the vortex pair moves as a unit in a fluid otherwise at rest, in the direction shown. Since, by definition, no flow crosses a streamline; streamlines in inviscid flows can be replaced by solid walls. If a solid wall is placed at the central streamline of the figure 5.28(a), the vortices will start moving opposite in direction to the earlier one. This analysis indicates that the stator hub vortex will move radially up on the pressure surface and down on the suction surface, which concurs with the observation at the exit of the rotor blade as shown in figure 5.28.

In the two test cases considered in this chapter, the interaction of the stator blade wake, the hub and the casing passage vortices with the rotor blade occur simultaneously. This makes it difficult to isolate the cause and the effect of a particular secondary flow on the rotor flow field. The interaction of only streamwise vortices with the downstream blade row is investigated in chapter 6.

5.10 Conclusions

The development of the steady and the unsteady three-dimensional flow in the turbine 2 has been described. The flow visualisation experiments, static pressure distributions and the measurements at the blade exit indicated that the secondary flows at the hub and the casing in the stator and rotor blades are lower from lower endwall blade loading. The lower endwall loading and the three dimensional stacking of the blades resulted in the reduced strength of the hub and casing passage vortices of the stator and the rotor. The passage vortices are also distributed over a large blade span.

The presence of the stator secondary flow at rotor exit reduced the rotor secondary flow development. The strongest rotor secondary flow occurs in the absence of the stator flow at rotor exit. The transport of the stator passage vortices inside the rotor blade is slightly different to the model proposed for the radially stacked blades. The pressure leg of the stator passage vortex was radially displaced upwards and the suction leg was entrained into the rotor passage vortex.

The agreement between the experimental and the computational results is good. The need for modelling secondary flows such as shroud leakage flows is also highlighted. Unsteady numerical simulations were found successful in accurately predicting the flow near the regions of secondary flow interaction. Comparisons between the steady and the unsteady numerical simulations with measurements highlighted the need for unsteady computations.

A method to calculate the loss generation rate in the blade row was developed from numerical simulations. The contribution of various regions of the blade row towards the total blade loss was evaluated from steady numerical computations. The maximum percentage of the loss was generated near the suction surface of the blade. A similar analysis was carried out using unsteady numerical simulations, where the entropy generation rate in each control volume was integrated over one stator passing period. The unsteady loss generation from both the stretching of the stator passage vortices in the rotor blade and the compression of the stator wake near the rotor throat in the mid-pitch region were demonstrated.

Delta Wing Vortex Transport in a High Pressure Steam Turbine Stage with Compound lean

6

6.1 Introduction

The steady and unsteady flow field of a three-dimensionally stacked high-pressure steam turbine blade was described in Chapter 5. The unsteady three-dimensional numerical simulations have shown that the wake and passage vortices transport, in the downstream blade row, generate additional losses. The interaction of the stator streamwise vortices with the downstream rotor blade row resulted in large variations in the rotor flow field. The earlier studies in chapters 4 and 5 were conducted in a stage environment where interaction of various forms of stator viscous flows (blade wake, hub and casing secondary flow) occur simultaneously with the downstream blade row. This makes it difficult to isolate the cause and effect of a particular secondary flow on the rotor flow field. In the present chapter, the interaction of the streamwise vortices with the downstream blade row has been investigated by shedding moving streamwise vortices into the downstream blade row using delta wings.

This chapter describes the vortex generation using delta wings and the characterisation of the vortex with angle of incidence and axial distance carried out in wind tunnel tests. This concept is extended and the delta wings are used in the rotating turbine rig to simulate the passage vortices of an upstream blade row. The flow field in the rotating turbine rig is investigated using miniature pneumatic probes and three axis hot-wire probes downstream of each blade row. This chapter also describes the steady and unsteady numerical simulations that are performed to further understand the flow field.

6.2 Delta Wing Tests in a Low Speed Wind Tunnel

The subsonic flow pattern over the top of a delta wing at an angle of attack is sketched in figure 6.1(a). The dominant aspects of this flow are the two vortex patterns that occur in the vicinity of the highly swept leading edges. These vortex patterns are created by the following mechanism. The pressure on the bottom surface of the wing at a flow incidence angle is higher than the pressure on the top surface. Thus, the flow on the bottom surface near the leading edge tries to curl around the leading edge from the bottom to the top. If the leading edge is sharp, the flow will separate along its entire length. This separated flow curls into a primary vortex, which exists above the wing inboard of each leading edge, as shown in figure 6.1(a). Figure 6.1(b) shows the smoke flow visualisation of the delta wing vortex in the present wind tunnel tests. The photograph was taken from behind the delta wing. Smoke was introduced through a smoke probe near the leading edge of the delta wing. Smoke introduced in the inlet boundary layer rolls up as a vortex while attempting to move from the bottom of the delta wing to the top of the delta wing. These vortices are strong and stable and can be used to simulate the passage vortices.

The aim of the present set of experiments is to demonstrate that the vortex shed from a delta wing is representative of a typical passage vortex. The characteristics of the delta wing vortex and its convection downstream were also investigated in the wind tunnel. The details of the delta wing geometry are given in figure 3.2. The trailing edge of the delta wing was designed to be sharp to avoid having a strong wake from the delta wing. The delta wing when mounted at an angle to the flow generates a pair of strong counter rotating vortices from its edges. As the aim of the present investigation is to understand the interaction of a vortex with the downstream blade, it was not necessary to release two counter rotating vortices. Instead, the delta wing was cut in the middle, fixed to the flat plate in the wind tunnel, thus producing a single vortex. In a turbine, the upstream passage vortex convects through the downstream blade passage, where the flow is accelerating in nature. The wind tunnel experiments were also carried out for understanding the vortex transport in an accelerating flow.

The delta wing was tested in a wind tunnel for two different configurations. One configuration produced a constant velocity flow, the other an accelerating flow. The test section layout is presented in figures 6.2 (a) and (b). Trip wires were used to ensure that the incoming boundary layer was turbulent in nature. A small gap between the wind tunnel exit and the test section was maintained such that the wind tunnel end-wall boundary layer was bled off. The delta wing was fixed to the flat plate at a distance from the leading edge where the boundary layer was similar to that entered in the rotating turbine ‘Peregrine’ test rig. For simulating the accelerating flow, the flat plate was rotated with respect to a hinge near the leading edge of the plate and locked at the other end of the flat plate with the wind tunnel wall. A false bottom was made for the test section such that it simulated the convergent area in the test section as shown in figure 6.1(b). Area traverses were carried out behind the delta wing at various axial locations for both test configurations.

Measurements were carried out with a five-hole pneumatic probe, for various angles of the delta wing with the incoming flow. Figure 6.3 presents the results from one such area traverse located at 60mm downstream of the delta wing. In this constant velocity test configuration, the delta wing was fixed at an angle of 10^0 to the flow. Figure 6.3(a) shows the contours of the coefficient of stagnation pressure loss (Y). The reference total and reference static pressures were measured upstream of the delta wing using a pitot-static pressure probe. The absolute stagnation pressure loss is non-dimensionalised with respect to the inlet dynamic pressure. A large loss core can be observed corresponding to the vortex with a maximum loss at the centre of the vortex. The corresponding secondary velocity vectors are presented in figure 6.3(b). The secondary flow in the present case is defined as the flow that is not in the main flow direction. Figure 6.3(c) presents the axial vorticity contours at the same measurement plane. These also indicate a large counter rotating vortex and the presence of irrotational flow around the vortex.

Figure 6.4 presents the stagnation pressure loss coefficient contours at three axial locations downstream of its generation. The measurement area is identical in all the three area traverses. Figure 6.4(a) corresponds to 20mm axial distance behind the delta wing and shows the vortex to be a concentrated region in the middle of the measurement plane. The second plot (figure 6.4(b)) was measured a further 20mm downstream of the first figure. It shows that the dissipation of the delta wing vortex has occurred as it now fills almost half of the measurement window. After an additional 60 mm, the vortex can now be observed in figure 6.4(c) to cover whole width of the measurement plane.

Similar experiments were carried out for the accelerating flow configuration at different delta wing incidence angles and at various axial locations. Static pressure tappings at various axial locations of the flat plate were provided to evaluate the isentropic velocity variation along the plate. Figure 6.5(a) presents the variations of the isentropic velocity ratio along the flat plate for the two test configurations. The velocity ratio is almost constant for the constant velocity configuration except near the delta wing region. This local variation in velocity is due to the stagnation of the flow while approaching the leading edge of the delta wing and immediate acceleration of the same. The corresponding velocity ratio distribution for the accelerating flow test case show that the velocities were increasing continuously with axial distance except near the delta wing. This local variation in velocity is again due to the stagnation of the flow while approaching the leading edge of the delta wing and immediate acceleration of the same. The area contraction ratio for this test configuration is fixed as 0.33 over a length of 910mm, which is the length of the plate.

Measurements were carried out with a five-hole pneumatic probe, for various angles of attack of the delta wing with the incoming flow. The reference total and reference static pressures were measured 160mm upstream of the delta wing using a pitot-static pressure probe. The absolute stagnation pressure loss is non-dimensionalised with respect to the inlet dynamic pressure. To compare different test conditions such as at different delta wing incidence angles and axial distance traverses, it is useful to integrate the loss over the traverse plane. The measurement window was designed such that it contains all of the loss generating regions. The measured stagnation pressure loss is mass weighted and corrected for the same mass flow for all test cases. The correction for the same mass flow was essential because, although the measurement window area is same for all the test cases, the mass flow increases with axial distance due to flow acceleration in the accelerating flow case. The correction is calculated as follows:

$$\Delta P_{0\,corr} = \frac{\dot{m}_i \Delta P_0 + (\dot{m}_{ref} - \dot{m}_i)*0}{\dot{m}_{ref}} \qquad (6.1)$$

where, $\dot{m}_{ref}$ is the reference mass flow, $\dot{m}_i$ is the mass flow of the measurement window and ΔP_0 is the stagnation pressure loss measured in a test condition.

Figure 6.5(b) present the variations of integrated loss coefficient with axial distance for both the constant area tests and accelerating flow tests. The measured loss is presented in two different forms. One is the stagnation pressure loss at the plane of measurement and the other is the total loss for the test condition including the constant area mixing losses. There is a very small increase in loss between the plane at 20mm and the plane at 100mm axial distance for constant velocity configuration. This is due to the dissipation of the vortex as shown in figure 6.4. The difference between the mixed out loss and the measured loss is also almost constant. This indicates that either the measurement grid resolution is not sufficient or there is only a small change in mixing loss with axial distance. The mixed out loss remained almost constant irrespective of the measurement location.

The situation is different in the accelerating flow tests. The measured loss has increased from 20mm behind the delta wing to 125mm behind the delta wing. The increase in loss is more than the constant area case. The loss coefficient in both the test cases is defined with respect to the inlet dynamic head so as to have a common reference for all the test configurations. The difference in the value of loss coefficient between the two test configurations at 20mm axial distance is due to the different inlet dynamic head between the test cases. The accelerating flow configuration has lower dynamic head and hence the higher loss coefficient. The total loss coefficient, which includes the mixing loss, follows the trend of measured loss from a value of 0.16 at 20mm axial distance to 0.23 at 125mm axial distance. The reason for the increase in loss is considered to be due to the additional loss generation as the vortex convects downstream. If the vortex is considered to be a streamtube, as the flow passes through a convergent area the flow accelerates resulting in the reduction of the stream-tube diameter. From Kelvin's theorem, the circulation around a stream-tube remains constant and so if the diameter of the tube is reduced by stretching, the streamwise vorticity is amplified. When a vortex is stretched along its axis, it can be shown that its secondary kinetic energy will vary as the square of its length (Denton (1993)). Hence, stretching a vortex will amplify its secondary kinetic energy. When this secondary kinetic energy is dissipated by viscous effects to a uniform flow, it will generate additional loss. The accelerating flow configuration tests indicate that the increase in loss with the downstream axial distance is significant. The difference between the mixed out loss and the measured loss in this case is also almost constant instead of reducing with axial distance. This indicates that either the measurement grid resolution is not sufficient or the measurement window does not capture all the loss generated in the delta wing vortex. It can also be due to the lower rate of mixing associated with the vortex.

Figure 6.6 presents the variation of circulation and loss coefficient with the angle of incidence for the accelerating flow configuration at 60mm axial distance downstream of the delta wing. As the angle of delta wing incidence increases, the pressure difference between the lower surface and the upper surface increases and results in both a larger vortex and higher circulation around the vortex. It can be observed in this figure that the circulation has increased linearly while the corresponding stagnation pressure loss increased non-linearly. One reason being the kinetic energy associated with the vortex scales with the square of the velocity, while the circulation scales with the velocity. The wind tunnel experiments successfully demonstrated the use of delta wings in representing a vortex. These delta wings are used in the low speed rotating turbine rig to simulate the passage vortex and its interaction with the downstream blade row. These experiments are discussed in the following section.

6.3 Delta Wing Flow in the Rotating Turbine Rig

Following wind tunnel testing, the 'half-delta wings' were fixed to the rotating hub of a single stage low-speed rotating turbine upstream of the stator blade row. The delta wing vortex transport in the downstream blade rows was studied in this experiment. Measurements were carried out at the exit planes of the delta wing row, the stator row and the rotor blade row, using pneumatic and slanted hot-wire probes. The details of the delta wing geometry are given in figure 3.2. The delta wings were fixed to the hub at an angle of 85^0 to the axial direction. This angle was required as the relative inlet flow angle to the blade row varies from 74^0 at the hub to 70^0 at the tip of the delta wing. An incidence of 10^0 is needed to simulate a representative passage vortex with the delta wing.

A comparison between the vortex generated by the present delta wing and a typical rotor passage vortex in a rotating turbine environment is presented in figure 6.7(a). The comparisons are made with the help of axial vorticity contours and some parameters as listed in the figure. As the measurements for the three-dimensionally stacked turbine were not available at the time of delta wing design, the passage vortex of the turbine investigated in chapter 4 was used in the delta wing design.

Figure 6.7(a)(i) presents the predicted contours of the axial vorticity at the exit of the rotor. The corresponding predicted contours for the present delta wing at the exit of the delta wing row is presented in figure 6.7(a)(ii). The contour levels for both the plots were kept the same for easy comparison. A passage vortex can be observed near the rotor hub in figure 6.7(a)(i) rotating in anti-clockwise direction and has a maximum vorticity of 3000 rad/sec at the centre of the region. It has a maximum velocity deficit of 20% of the freestream value with a diameter of 47.8mm. The results at the delta wing exit are presented in two rotor passage widths, as the present configuration has 21 delta wings in comparison to 42 rotor blades, for the reasons explained later in the section. The delta wing vortex can be observed in figure 6.7(a)(ii) rotating anti-clockwise direction with a maximum vorticity 3000 rad/sec in the centre of the region. It can be observed that the size of the vortex and velocity deficit in the centre of the vortex also compares well with the rotor vortex. This indicates that the delta wing vortex is a good simulation of a rotor passage vortex at the hub.

Figure 6.7(b) presents the three dimensional solid model of the turbine rig showing the delta wing row, the stator and the rotor with shroud arrangement. Initial experiments with 42 delta wings (equal to the number of rotor blades) upstream of the stator were not successful. This was due to the large amount of blockage created by the delta wings because they have a stagger angle of 85^0, which is almost perpendicular to the axial direction. After several trials, it was decided to use 21 delta wings instead of 42, resulting in less mass flow blockage and associated flow distortion.

Modelling this delta wing with a structured CFD is a very challenging problem due to the grid skew of 85^0 to the axial direction. Extreme care has to be taken in terms of selection of various control parameters, as these will be very crucial in getting an accurate solution. The computational mesh for the delta wing and the stator and rotor blade rows is shown in figure 6.7(c). Figure 6.7(d) shows the detail of the same mesh near to the stator leading edge. The large skew in the delta wing mesh can be seen in both the figures. Delta wing was represented in the computations as a blade of varying chord and thickness with the height of the wing.

As there was no access for rotating traverse gear at the measurement location behind the delta wings, relative frame traverses were not carried out. Instead, radial traverses and area traverses were carried out in the absolute frame of reference with a five-hole probe to quantify the loss generated due to the delta wing vortex. Figure 6.8 shows the secondary velocity vectors at the exit of the delta wing measured using a single slant hot-wire (plane 0, 23% of stator C_x before the stator leading edge). The single slant hot-wire technique is described in detail in section 3.5.2. A much denser measurement grid is used near the vortex for better resolution of the delta wing vortex. The time variation of the measured velocity is expressed as rotor blade pitch for visualising the delta wing exit flow. A vortex can be identified at 17% blade span rotating in counter clockwise direction in figure 6.8. The vortex covers from 5% to 25% of blade span with a centre at 17% blade span. It was shown in chapters 4 and 5 that the turbulence intensity is a good indicator of the loss generating regions. Figure 6.9(a) presents the measured rms intensity data behind the delta wing. It is defined as follows:

$$\mathrm{rms} = \sqrt{\ll \alpha(t)^2 \gg} = \sqrt{\tfrac{1}{N}\sum_{n=1}^{N}(\alpha(t,n) - <\alpha(t)>)^2} \,/\, \overline{\alpha} \qquad (6.2)$$

The ensemble root-mean-square (rms) represents the amount of deviation, positive or negative, from the average velocity value at that phase. The rms intensity corresponding to the delta wing vortex can be seen to be as high as 9.1% at 15% blade span. The contours of the relative stagnation pressure loss coefficient (Y, see equation 3.18 for definition), from the steady numerical simulations carried out with MULTIP99, are presented in figure 6.9(b) at axial plane 0. The loss core corresponding to the delta wing vortex can be observed at 18% blade span. The predicted loss at the centre of the vortex has a deficit of 35% dynamic pressure.

Figure 6.10 (a) presents the axial vorticity contours from hot-wire measurements while the corresponding predictions from CFD are given in figure 6.10(b). The positive values of vorticity correspond to fluid rotating in clockwise direction and negative values of vorticity indicating the counter-clockwise rotation. The core of delta wing vortex can be identified by the region where the vorticity has a value of -1500 radians/sec in the measurements while the CFD predicts this to be -3000 radians/sec. It was mentioned in chapter 3, the necessity to increase the blade speed in numerical calculation from 550 rpm to 1125 rpm. As the vorticity and angular velocity are related, the true predicted vorticity in this region is also around -1500 radians/sec, which matches well with the measurements. On the top of the delta wing vortex, there is a region of counter rotating vorticity. The origins of this have to be investigated further. The rest of the flow field can be observed to have very nearly zero vorticity, except in the endwall boundary layers.

The results from the radial traverses carried out with a five-hole pneumatic probe behind the delta wing row are presented in figure 6.11 and are compared with predictions from numerical simulations. Figure 6.11(a) shows the pitchwise averaged spanwise variations of the absolute yaw angle. The points are from measurements and the solid line is from the predictions. The overturning near the hub end wall and underturning towards the mid-span, indicating a classical vortex pattern, can be observed at around 10% blade height. The large variations in yaw angles indicate that the strength of the vortex is very high. There is a good agreement between the numerical predictions and the measured data except at around 10-20% span. The computation predicts a larger delta wing vortex compared to the measurements. This difference in the size of the delta wing vortex may be due to the difference in predicting the mixing of the vortex with the free-stream flow. This may be due to the use of a higher than normal artificial viscosity coefficient, which was essential in stabilising the calculation especially with an 85^0 skew in the computational grid.

Figure 6.11(b) shows the outlet to inlet axial velocity density ratio both from measurements and simulations. This ratio essentially describes the spanwise distribution of the mass flow and its integrated value accounts for the change of mass flow from the inlet to the outlet. There is a very close agreement between the measurements and computations except near the vortex centre. There is a large reduction in mass flow passing through the hub region up to 20% blade span, indicating that the blockage created by the presence of delta wing. This is the only difference between the flow conditions behind a passage vortex associated with a rotor blade row and the simulation of the passage vortex with a delta wing row. With no other alternatives available, it was decided to continue the experiments and analyse the data keeping this point in mind.

Figure 6.11(c) presents the variations of stagnation pressure loss coefficient (Y, see equation 3.18 for definition) at the same measurement plane. The stagnation pressure loss is high in the region upto 15% span, corresponding to the vortex. The difference between the loss measurements and predictions (from 15-25% span) confirms that the predicted loss core is more diffuse than the measured. Except at this region, there is a good agreement between the measurements and predictions throughout the blade span. The computational grid for this case is highly skewed due to the delta wing geometry. In spite of this limitation, very good agreement between the losses, yaw angles, mass distribution and vorticity indicate the robustness of the numerical simulation model used and gives confidence in using the calculations where a direct comparison is not available.

6.4 Stator Exit Flow

As the delta wing vortex convects through the downstream stator blade row, it interacts with it. The flow field at the exit of the stator is investigated with the help of measurements at plane 1, located at 8.4% of stator C_x distance downstream of the stator trailing edge in the absolute frame of reference. This measurement plane is different to the datum configuration, which is at 13% C_x downstream of the stator trailing edge but same as the 'rotating hub' configuration. Hence, a direct comparison between the 'rotating hub' and 'delta wing' configurations is possible. Area traverses at plane 1 were performed using a 5-hole probe for steady flow field and a three-axis hotwire for the time resolved flow field. The measurement techniques and the probe details were outlined in section 3.5.

6.4.1 Steady Stator Exit Flow Field - 'Delta wing' configuration

The contours of stagnation pressure loss coefficient (Y, see equation 3.18 for definition) measured with a five-hole probe are presented in figure 6.12(a). The loss regions due to the blade wake can be identified in the middle of the plot. In addition to blade wake, there is a loss core near the casing due to the endwall secondary flow. Near the hub, a large loss core due to the passage vortex covering almost 60% of the blade pitch extending upto the pressure surface side of the blade (region A) can be observed. A comparison of the contours of the loss coefficient between the rotating hub case in figure 5.25(a) and the present indicated that the hub passage vortex in this test case is large and has higher loss. The delta wing vortex interaction with the stator blade has increased the strength of the hub passage vortex and hence the increase in loss associated with the vortex.

Figure 6.12(b) presents the predicted contours of the stagnation pressure loss coefficient (Y) from the steady numerical simulations. The axial location and the contour intervals are the same as for the measured data. It can be observed that the loss contours agree well with the measurements except near the region corresponding to the hub passage vortex. In this case, the loss core corresponding to the hub passage vortex can be observed at a higher radial location of 30% blade span instead of at 10% blade span. The size and the strength of the loss core are also much larger than in the measurements. It was observed in section 5.7 while comparing the flow field at the exit of the rotor that the steady computations did not accurately predict the unsteady transport phenomena of the stator flow. A similar explanation can be given here for the difference between the measurements and the predictions at stator exit. The flow at this location is a combination of the interaction between the delta wing vortex and the stator blade row.

Figure 6.12(c) presents the phase averaged time mean turbulence intensity ($\overline{Tu}$, see equation 5.8 for definition) measured using a three axis hotwire. It can be observed that the turbulence intensity contours are in good agreement with the loss coefficient contours at various viscous regions in the flow like the blade wake and the passage vortices. The maximum turbulence intensities of 11.7% can be seen in the centre of the vortex as compared to around 8% in the centre of the wake and 1.8% in the free stream regions. The passage vortex region near the casing has lower turbulence intensity levels of 8% compared to the hub region. The turbulence intensity on the pressure side of the blade (region A) is around 4% and coincides with the loss region as noticed in figure 6.12(a).

Another parameter, which characterises the flow, is the yaw angle. The measured yaw angles of the flow at the stator exit along with the secondary velocity vectors are presented in figure 6.13. Although it was observed from the loss coefficient predictions that there are differences between the steady computations and the measurements especially near the hub vortex region, still a comparison has been made between the same to understand how close the steady computations can predict the flow field in figure 6.13. Figure 6.13(a) presents the contours of the absolute yaw angle from five-hole probe measurements. The flow overturning caused by the strong hub secondary flow at suction side corner can be observed so is the flow overturning near the casing. The overturning near the hub can be seen up to 10% blade span while the flow underturning can be observed from 10 % to 25% of blade span. This whole region corresponds to the loss core as shown in figure 6.12(a) associated with the strong hub passage vortex. This underturning can be seen extending to the pressure side of the blade.

The results from steady simulations are given in figure 6.13(b). The results from unsteady simulations are discussed in detail in section 7.2. The overturning of the flow caused by the secondary flow near the casing can be observed. The overturning near the hub can be observed from hub to 18% blade span while the flow underturning can be observed from 18% to 30% blade span. Though the steady CFD does not model the vortex transport in the downstream blade row, it is able to predict the underturning up to 30% span. There is also a discrepancy in the location of the hub passage vortex between the steady CFD and measurements.

The secondary velocity vectors give useful information about the nature of the secondary flow. Figure 6.13(c) presents the secondary velocity vectors calculated from the measured data. Near the hub, a strong secondary flow pattern occupying almost 20% blade span is observed. The casing secondary flow is weaker than the hub and covers up to 20% blade span from casing. Another vortical flow is also observed near region A rotating opposite in direction to the hub secondary flow at 20% blade span. This vortical flow is further discussed in section 6.4.2 with the help of the unsteady measurements. Figure 6.13(d) presents the corresponding calculated secondary velocity vectors from steady CFD. Again, the hub passage vortex can be observed at a higher radial location than the measurements.

The pitchwise averages of the flow field measured at the exit of rotor are given in figure 6.14. Figure 6.14(a) gives the comparisons of the stagnation pressure loss coefficient for the datum, rotating hub and delta wing test configurations. The reference stagnation pressure for all the measurements is the mean stagnation pressure measured at the inlet of the rig at mid radius location. The stagnation pressure loss across the blade row is non-dimensionalised with the exit dynamic head behind the stator. The definition of the loss coefficient is given in equation 3.18. The loss behind the stator for 'delta wing' configuration includes the loss from delta wing. The inlet flow angle to the delta wing varies from 74^0 at hub to 70^0 at the tip of the wing. The exit flow angle from delta wing calculated from the simulations is 73^0 indicating very little or negligible flow turning in the delta wing. This indicates that there is no work input to the flow from the delta wing.

The local increase in loss coefficient (Y=0.032, figure 6.14(a)) corresponding to the passage vortex region can be seen near 10% blade span for the datum configuration. The loss coefficient corresponding to the passage vortex at the hub increased to 0.054 for the rotating hub configuration and it is located at 15% of the blade span. A larger increase in loss coefficient to 0.11 for the delta wing configuration is observed at 15% span. Some of this loss is due to the delta wing vortex and some due to the interaction between the delta wing vortex and the stator blade row and these are further discussed in the later part of this section.

Figure 6.14(b) presents the pitchwise averages of the measured flow angle data for the three test configurations. The flow overturning near the hub region and underturning towards the mid-span, indicating a classical vortex pattern, can be observed at around 10% blade height for the datum test configuration. The maximum variation of the yaw angle from the mean flow angle is increased for the rotating hub configuration, indicating a stronger vortex than the datum test configuration. The increase in loss and the upward movement of the location of the passage vortex for the rotating hub case confirms the findings of Bindon (1980), Boletis *et al.* (1983), Walsh and Gregory-Smith (1987). The increase in secondary flow and the upward movement of the vortex is due to the skew in the inlet boundary layer to the stator. The yaw angle variation is larger for the delta wing configuration indicating increased strength of the passage vortex compared with the other two test configurations. At all other locations from 40% blade span to casing there is little variation in the flow angle variation between the three configurations.

Table 6.1 presents the area-integrated values of loss coefficient and yaw angle for the three test cases. All the integrated values are evaluated from area traverses carried out with a five-hole pneumatic probe. The uncertainty associated with the measurement of loss coefficient in the stationary frame of measurements is given as ± 0.0015. Further details of the uncertainty analysis are given in Appendix A1. All the loss coefficients are defined with respect to the rig inlet conditions. The integrated loss coefficient behind the stator for the datum case is 0.0210, while the loss coefficients for the rotating hub and delta wing configuration are 0.0246 and 0.0375 respectively. The integrated loss coefficient measured behind the delta wing is 0.0077. This loss magnitude is 36.7% of the datum loss. The difference in losses between the datum and rotating hub configuration can be attributed to the increase in loss due to the skewed incoming boundary layer and is calculated as 17.1% of the datum loss. The difference in loss between the delta wing and the datum test configuration can similarly be evaluated as 78.6% of the datum stator loss. After considering the contribution of delta wing loss (36.7%) and the loss due to skewed inlet boundary layer (17.1%), the additional loss generated in the stator is 0.0052 or 24.8% with an error of $\pm 7.14\%$ of the datum stator loss. This additional loss is due to the interaction between the delta wing vortex and the downstream stator blade and also include the delta wing mixing loss. The integrated results of the loss show that the additional loss generated in the stator blade is significant. The unsteady stator flow field is further discussed in the following section and the results from unsteady numerical simulations are discussed later in sections 7.2 and 7.4.

6.4.2 Unsteady Stator Exit Flow Field - 'Delta wing' configuration

It has been shown in the previous section that there is a difference between the steady numerical predictions and the experiments indicating the effect of unsteadiness on the stator flow field. This unsteadiness arises due to the relative motion between the delta wing vortex and the stator row. The work presented in this section will give an insight into the mechanisms of interaction between the delta wing vortex and the stator blade. Extensive time resolved data has been obtained downstream of the stator (plane 1, 8.4% of stator C_x downstream of stator trailing edge) using a three axis hot-wire probe in an absolute frame of reference. The measurement plane is at the same location as that of the 'rotating hub' test case and also at the same location as that of the five-hole probe measurement plane discussed in section 6.4.1. The method used for the data acquisition and data reduction is outlined in section 3.5. The present 'delta wing' test configuration has 21 delta wings, in comparison with 42 rotor blades. Hence, one appearance of the delta wing vortex behind the stator can be expected over two-rotor blade passing periods.

Figure 6.15 presents the contours of the instantaneous turbulence intensity (Tu, see equation 5.8 for definition) in two rotor passing periods behind the stator (plane 1) from three axis hotwire measurements. The stator wake, the hub and the casing passage vortices can be identified by the high-turbulence intensity regions, as marked in the figure. The maximum turbulence intensities correspond to the passage vortex at the hub and in the intersection between the wake and the vortex. The turbulence levels are as high as 11.7% in the centre of hub passage vortex compared to 7% in the centre of the wake. It can be observed that in the regions corresponding to hub passage vortex, the turbulence intensity varies with time during the two-rotor wake passing periods. This region has a minimum value of 9.9% at t/τ=0.750 and a maximum value of 11.7% at t/τ=1.875.

The structure of the stator secondary flow is similar at all instants of time with little variations from one time instant to another. The instantaneous contours of the yaw angle are shown in figure 6.16 at the same measurement plane 1 in two rotor passing periods. The flow overturning near the hub and casing endwalls, and the flow underturning just in board of these regions, which are characteristics of streamwise vortices, can be observed at all instants of time. The flow underturning has a minimum value at t/τ=1.875 and a maximum value at t/τ=0.375. These two time instants coincide with the maximum and minimum turbulence intensity values as shown in figure 6.15. There is no indication of any other passage vortex near the hub from the yaw angle data or the pitch angle data (not presented in the thesis). The turbulence intensity results combined with the yaw and pitch angle data suggests that the delta wing vortex has mixed with the stator passage vortex at the hub, hence cannot be identified as a separate entity. Nevertheless, the presence of the delta wing vortex inside this region is confirmed using frequency spectrum data presented later in the section and also from the results of the unsteady numerical simulations presented in section 7.2.

6.4.3 Spectrum Analysis at Stator Exit

Spectral analysis have been carried out at the present measurement location (plane1) by recording the velocity from three wires of a hot-wire probe over a long period of time. The data has been sampled at various important locations in the area traverse such as the centres of the wake, hub and casing passage vortices and in the free stream. The data is logged at a frequency of 10KHz to cover the range of the frequencies of interest. A total of 4096 points were acquired after the once per revolution signal. The spectral analysis was carried out using a Fast Fourier Transformation (FFT) of the tangential velocity data. The spectral analysis of the axial and radial components of the velocity also resulted in similar conclusions and hence only the tangential velocity data (this component being dominant in the total velocity) is presented here.

Figure 6.17 presents the results from the spectral analysis for the ‘rotating hub’ test configuration at 8.4% C_x downstream of the stator trailing edge (plane 1). The starting point of the arrows pointing to the spectra shows the measurement location. At each location, both the frequency spectrum and a time trace of the tangential velocity are given. The velocity trace is given for only four blade-passing periods. In the freestream (figure 6.17(a)), the predominant frequency is 391 Hz with a power spectral density of 3640 m^2/sec (psd). This frequency very close to the downstream rotor blade passing (385Hz) and results from the potential interaction of the downstream rotor blade with the upstream stator flow. In the centre of the wake (figure 6.17(b)) the spectrum still shows the rotor blade passing frequency but with a lower power spectral density of 456 psd. This is an order of magnitude less than the freestream value indicating the reduction in magnitude of the contribution from the blade passing frequencies inside the wake. Figure 6.17(c) shows the spectrum inside the stator passage vortex near casing. No predominant frequencies can be observed at this location. Similar conclusions can be drawn from the data near the hub passage vortex as shown in figure 6.17(d). To summarise, these spectra clearly demonstrate the reduction of the power spectral density corresponding to the blade passing frequencies in the viscous regions.

Figure 6.18 presents the spectrum analysis carried out behind the stator (8.4% C_x downstream of the stator trailing edge, plane 1) for the 'delta wing' test configuration. The results in the freestream of the stator (figure 6.18(a)) indicate two predominant frequencies, one at 195 Hz with 526 psd, and the other at 385Hz with 2300 psd. The downstream rotor blade passing frequency in this case also is 385Hz, and the upstream delta wing blade passing frequency is 192.5Hz. The two predominant peaks in the frequency spectrum correspond to these two potential waves; one due to the downstream rotor and the other due to the upstream delta wing. Inside the wake, the power spectral densities corresponding to these blade-passing frequencies are reduced as shown in figure 6.18(b). The frequency corresponding to the delta wing is not predominant, while the frequency corresponding to the rotor is reduced with a power spectral density of 575 psd. The results inside the casing passage vortex suggest that there are no predominant frequencies observed at this location (figure 6.18(c)). The spectrum in the centre of the hub passage vortex is different (figure 6.18(d)). There are two dominant frequencies at this location. One at 193 Hz with 1300psd and the other is at 391Hz with 381 psd. These frequencies again correspond to the delta wing and the rotor passing frequencies respectively. At the corresponding location for the rotating hub case, there are no dominant frequencies observed. This suggests that the delta wing vortex is responsible for the dominance at these frequencies. This indicates that the vortex near the hub is a result of the combination of the stator hub passage vortex and the upstream delta wing vortex.

6.4.4 Reynolds stresses at Stator Exit

It is known that the major contributor to the loss production in turbulent boundary layers and passage vortices is the 'kinetic energy cascade' from larger eddies to smaller eddies. These smaller eddies in turn through turbulent dissipation generate energy losses. It is therefore possible to study the loss generation in the stator blade row by measuring the turbulence production in the blade. It is possible to calculate the Reynolds stresses using the time-resolved hotwire data, which with velocity gradients in the flow describe the turbulence production rate in the blade row.

Figure 6.19 presents the phase averaged time mean Reynolds shear stress contours at the exit of the stator. The perturbation $\overline{V_x' V_r'}$ represents the tangential component of the Reynolds shear stress, $\overline{V'_\theta V'_r}$ represents the axial and $\overline{V'_\theta V'_x}$ represents the radial component of the Reynolds shear stress. The tangential component of the Reynolds shear stress can be calculated as

$$\overline{V'_x V'_r} = \frac{1}{\tau}\sum_{t=0}^{\tau}\left\{\frac{1}{N}\sum_{n=1}^{N}\left(V_x(t,n)-V_x(t)\right)\left(V_r(t,n)-V_r(t)\right)\right\} \tag{6.3}$$

where 'n' represent the ensemble number in a total of 'N' ensembles and time 't' is measured from a once-per-cycle datum point for a periodic process. The velocity $V_x(t)$ is the ensemble mean of the axial velocity at a point in the measurement grid. Similarly, the other two Reynolds shear stresses are also defined. The Reynolds stress components does not include any periodic terms as the ensemble mean is removed from the velocity.

The highest values of these stresses occur near the casing and the hub secondary flow regions and in the blade wake regions. These are also the regions of large velocity gradients. Figure 6.19(a) shows the tangential component of the shear stress with a maximum value of 1.6 in the centre of the hub passage vortex region, and -0.4 in the centre of the casing passage vortex. Figure 6.19(b) shows the axial component of the shear stress with a value of -0.8 and 0.2 in the centre of the hub and casing passage vortices respectively. The fact that the Reynolds shear stress $\overline{V'_\theta V'_r}$ values are positive at the casing and negative in the hub region indicates the two counter-rotating vortical structures in the flow. The magnitude of the shear stress at hub is large compared to the casing, indicating the stronger hub passage vortex compared to the casing.

Figure 6.19(c) presents the radial component of the Reynolds shear stress ($\overline{V'_\theta V'_x}$) behind the stator blade row. This component can identify the shear in the blade wakes. Two different regions, one corresponding to suction surface (positive shear) and the other due to pressure surface (negative shear) can be observed on the suction side and the pressure side of the stator blade respectively. These are due to the boundary layers on the respective blade surfaces. The value of the shear stress on the pressure side is around -2.0 as compared to 0.6 on the suction side, indicating that the pressure side boundary layer shear is stronger than the suction side. This is because of the thin boundary layers on the pressure side of the blade compared to the suction side. It is also to be noted that the magnitudes of radial and streamwise stresses are not of equal magnitude.

The three individual components of the Reynolds normal stress in x, θ and r directions were also calculated. The contours of these Reynolds normal stress components are identical in shape to the turbulence intensity (Tu) data given in figure 6.15, but with different magnitudes. Hence, only the pitchwise averaged values of these components are presented in figure 6.20. Figure 6.20(a) shows the pitchwise averaged spanwise variations of Reynolds normal stresses. The local increase in the normal stresses corresponding to hub and casing passage vortices can be observed at around 15% and 90% blade span respectively. In isotropic turbulence, the Reynolds normal stresses in x, r, θ directions would be equal. We can define non-isotropy as the deviation of the normal stresses from being equal. In figure 6.20(a), it can be observed that the radial component of the normal stress is much less compared to the axial and tangential components. The maximum value of the Reynolds normal stress corresponds to tangential direction at this plane. This shows that the turbulence in the flow is non-isotropic.

The Reynolds shear stress data discussed earlier was pitchwise averaged and presented as spanwise variations in figure 6.20(b). The magnitudes of the shear stress are greatest near the spanwise location corresponding to hub and casing secondary flows at 15% and 90% blade span. At other spanwise locations, the magnitudes are smaller. The Reynolds stresses multiplied with the velocity gradients in the flow represents the turbulence production terms in the flow. These turbulent eddies in turn through turbulent dissipation generate energy losses. The evaluation of the Reynolds normal and shear stress terms facilitated the identification and evaluation of loss generating regions in the blade exit flow. These Reynolds stresses can be useful as test data in the development of advanced turbulence models.

6.5 Rotor Exit Flow

It has been shown in section 6.4.1 that the interaction between the delta wing vortex and the stator blade row resulted in very strong stator passage vortex at the stator hub. It covered almost 60 to 75% of the blade pitch and up to 20% of the blade height. This stator passage vortex interacts with the downstream rotor blade and generates a complex flow field in the rotor blade row. The flow field at the rotor exit is discussed with the help of measurements at plane 3, located 10% of rotor C_x distance downstream of the rotor trailing edge, in the relative frame of reference. Area traverses at plane 3 were performed in the relative frame using a five-hole probe for steady flow field measurements and a three axis hot-wire for time resolved flow field data. The measurement techniques and the probe details are outlined in section 3.5.

6.5.1 Steady Rotor Exit Flow Field - 'Delta wing' configuration

The results at the exit of the rotor (plane 3, 10% C_x from blade trailing edge) from five-hole probe, three axis hot-wire measurements and numerical simulations are presented in figure 6.21. This measurement plane is different to the datum configuration, which is at 26% C_x downstream of the stator trailing edge but same as the 'rotating hub' configuration. Hence, a direct comparison between the 'rotating hub' and 'delta wing' configurations is possible. Figure 6.21(a) presents the contours of relative stagnation pressure loss coefficient (Y, see equation 3.18 for definition) from five-hole probe measurements and the corresponding results from numerical simulations are shown in figure 6.21(b). The rotor wake can be identified as a thin loss region in the centre of the area traverse. The loss cores corresponding to the hub and the casing secondary flow and shroud leakage flow are found at 12%, 73% and 95% blade span respectively. In addition to these features, another loss core can be identified to the right of the hub secondary flow occupying the entire blade pitch (region A). There is another loss core observed on the pressure side of the blade near casing region and denoted by B. Both these regions are further discussed with the help of additional experimental data in the later parts of the section.

A comparison of the contours of the loss coefficient between the rotating hub case in figure 5.26(a) and the present case indicated that the rotor flow field is very similar including the magnitudes of loss. The loss corresponding to the rotor hub passage vortex is lower in magnitude in 'delta wing' case (Y_{max}=0.325) than the corresponding rotating hub case (Y_{max}=0.390). The loss corresponding to the stator flow interaction region in this test case is much higher (Y_{max}=0.15) compared to the rotating hub case (Y_{max}=0.09). This indicates that the stronger stator hub passage vortex interaction with the rotor blade row has reduced the loss corresponding to the rotor passage vortex at the hub.

The results from numerical simulations are different to the measurements (figure 6.21(b)). The flow near the casing region is dominated by a secondary passage vortex. The experimental data is different, with the shroud leakage flow dominating most of the casing region. In the present simulation, the shroud leakage is not modelled in the calculation. This explains the difference between the predicted and measured casing secondary flows. Surprisingly, the hub secondary flow is entirely different to that measured. There is no indication of the loss core corresponding to the hub passage vortex. This shows that the steady computations in an unsteady flow situation like this can not be used. Wherein the interaction between the upstream stator flow and the downstream blade row is significant.

Figure 6.21(c) presents the contours of the phase averaged time-mean turbulence intensity ($\overline{Tu}$, see equation 5.8 for definition) measured with a three-axis hotwire at the same measurement plane. The turbulence intensity in the centre of the wake is 13%, with maximum (18%) in the centre of shroud leakage flow and hub passage vortex. The high turbulence region near the hub extends from suction side to the pressure side of the passage, occupying the whole blade pitch. The location of this region on the pressure side (35% blade span) is much higher than on the suction side (17% blade span) similar to the observed for region A in figure 6.21(a). The turbulence intensity in the free stream region is higher at 8% compared to the stator exit value of 1.8%. Hence, it can be said that the rotor free-stream turbulence has increased as a result of the interaction of the stator wake flow with the rotor flow field. The increase in turbulence intensity is also observed in the centre of the wake (13%) compared to the corresponding stator wake value of 9%. The overall results of turbulence intensity match well with the stagnation pressure loss contours from five-hole probe measurements.

A comparison of the contours of the turbulence intensity between the rotating hub case in figure 5.26(b) and the present case indicated that the rotor flow field is very similar including the magnitudes of turbulence intensity. A large turbulence intensity region can be observed in the present test case covering whole of the rotor blade pitch compared to a much smaller region in ‘rotating hub’ case. In both the test cases, the radial upward migration of the turbulence region near the pressure surface can be observed. The stator flow interaction with the rotor blade is further discussed with the help of unsteady measurements in section 6.5.2.

The secondary velocity vectors at the measurement plane 3 are plotted in figure 6.22 in relative frame of reference. The secondary velocity vectors are defined as the deviation from the mean flow angle for the whole area. Two vortices can be identified in figure 6.22 from the five-hole probe measurements. The clockwise rotating vortical structure can be observed at 10% blade span and in the middle of the plot. This vortex is due to rotor passage vortex while the other vortex which is rotating in the opposite direction (region A) can be observed at 20% blade span to the right of blade suction surface.

Figure 6.23 presents the contours of the phase averaged time-mean relative yaw angle from three axis hotwire measurements at the exit of the rotor (plane 3). The area averaged relative yaw angle at this location is -69.96 degrees. With respect to this, the overturned flow can be observed near the hub and casing regions due to the end wall secondary flow and the underturned flow towards the mid span. In addition to this, another region of overturned flow (region A) can be observed at 30% blade span occupying the whole of rotor pitch from suction surface to pressure surface. The corresponding region is identified with high loss and high turbulence intensities in figure 6.21 and a vortical flow counter rotating to the rotor hub vortex in figure 6.22. This suggests that this flow may be the time mean manifestation of the stator passage vortex. The origin of this region is further discussed with the help of unsteady measurements in section 6.5.2.

The pitchwise averaged spanwise distributions of the measured flow for two test cases are presented in figure 6.24 at the exit of the rotor (plane 3, 10% C_x downstream of the rotor trailing edge). The comparisons are not made with the datum test configuration as the measurement plane in that case is at 26% C_x instead of 10% C_x downstream of the rotor. Figure 6.24(a) presents the relative yaw angle distributions for the two test configurations: rotating hub and delta wing. The classical overturning near the hub endwall and underturning towards the midspan is seen in the case of a rotating hub configuration with a vortex at around 17% blade span. An entirely different phenomenon is observed for the delta wing test case. Large underturning near the hub region (up to 15% blade span) and overturning towards the mid-span region (from 20% to 40% blade span) can be observed for the delta wing case. This is similar to the results obtained by Sharma *et al*. (1988). The flow overturning near 30% span corresponds to the loss core (region A) in figure 6.21(a) and the vortex discussed in figure 6.22. The results from stator exit traverses show that the passage vortex behind the stator is very strong, occupies the whole stator pitch and has same sense of rotation as a classical stator passage vortex. The stator passage vortex while transporting through downstream blade produced an overturned flow at 30% span while in the ‘datum’ and ‘rotating hub’ configurations it did not result in overturned flow. This indicates that the flow overturning towards the mid-span is due to the interaction of the stator flow with the rotor and depends on the strength of the incoming passage vortex. At other spanwise locations, a good agreement between both test configurations from 90 to 100 % blade span can be observed indicating the little or negligible effect of delta wing vortex transport on the shroud leakage flow. The difference in yaw angle distribution from 40 to 90% blade span between the two configurations also indicate the effect of the stator vortex transport on the rotor flow field.

Figure 6.24(b) shows the outlet to the inlet ratio of the axial velocity density with blade span for both test configurations. There is a small difference between the two test cases from 40% to 100% blade span. A large deficit in mass flow distribution is observed corresponding to the overturned flow region of figure 6.24(a), indicating that this region has much lower axial velocities than the end wall region.

The pitchwise averaged variations of the relative stagnation pressure loss coefficient (Y) are shown in figure 6.24(c). The local increase in loss is observed corresponding to the secondary flow regions at the hub and the casing. A difference in magnitudes of loss between the two test cases is observed from the hub region to up to 38% blade span. The reduction in loss for the delta wing case from 20% span to 38% span can be observed and may be attributed to the reduction in rotor secondary flow in the delta wing test case.

Table 6.2 summarises the area-integrated values of the loss coefficient and the relative yaw angle for both the test cases. The uncertainty associated with the measurement of loss coefficient in the rotating frame of measurements is given as ± 0.002. Further details of the uncertainty analysis are given in Appendix A1. The integrated loss coefficient behind the rotor for the rotating hub configuration is 0.0691 and the corresponding value for the delta wing configuration is 0.0687. This reduction in loss coefficient is within the uncertainty limits. This loss coefficient data still indicate that the loss has remained constant if not reduced in this delta wing case in spite of having a strong incoming stator hub passage vortex.

6.5.2 Unsteady Rotor Exit Flow - ‘Delta wing’ configuration

Extensive time resolved data has been obtained downstream of the rotor (Plane 3, 10% of rotor C_x downstream of rotor trailing edge) using a three axis hot-wire probe in the relative frame of reference. These data are used to understand the interaction between hub passage vortex of the stator with the rotor row. The methods used for the data acquisition and data reduction were outlined in section 3.5. The present ‘delta wing’ test configuration has 21 delta wings, in comparison with 36 stator blades and 42 rotor blades. Hence, one appearance of the delta wing vortex behind the rotor can be expected over two-stator blade passing periods. The measurements are carried out for three rotor pitches. However, no variation between the two subsequent rotor passages is observed both in the measurements and in the unsteady simulations. The unsteady simulations are further discussed in section 7.2. Measurements over only one stator-passing period are shown in the thesis.

Figure 6.25 presents the contours of instantaneous turbulence intensity (Tu) at rotor exit (plane 3) over one stator wake passing period. The blade wake, hub and casing secondary flow, stator flow interaction regions can be observed with high turbulence intensities as shown in figure 6.25. The magnitude and size turbulence intensity corresponding to the stator interaction region (region A) varied periodically with time over one stator wake-passing period. Region A had a minimum value of 16.8% turbulence intensity at time $t/\tau=0.375$ and a maximum value of 20.4% at time $t/\tau=0.000$. This region occupied almost the whole of the blade pitch at 25% blade height. A comparison of the contours of the turbulence intensity between the rotating hub case in figure 5.28 and the present case indicated that the rotor flow field is very similar including the magnitudes of turbulence intensity. A large turbulence intensity region (region A) can be observed in the present test case covering whole of the rotor blade pitch compared to a much smaller region in ‘rotating hub’ case. The radial migration of the turbulence core on the pressure side is at higher radii than on the suction side of the blade, which is similar to the observed in figure 5.28. It has been shown from vortex dynamics in section 5.9 that the stator vortex moves radially downwards on the suction surface and upward on the pressure surface from the action of image vortices inside the blade surfaces. This is further discussed in section 7.3.

The size and magnitude of turbulence intensity varied little in the shroud region over one wake passing period. This suggests that there is a negligible effect of stator vortex transport on the unsteadiness of shroud leakage flow. Another region B located to the right of the rotor wake can be observed with higher turbulence intensity than the surrounding at time $t/\tau=0.50$. This region moves to the right towards the rotor wake and thickens the rotor wake width after quarter of stator passing period at $t/\tau=0.75$. The rotor wake movement continues to the right at time $t/\tau=0.875$ and the rotor wake reverts to a thin turbulence intensity region by time $t/\tau=0.00$. This periodic variation indicates that this region may be due to the transport of the stator wake in the rotor blade. The rotor blade wake can be distinguished from the rest of the secondary flow from 30% span to 75% span. In these spanwise locations the turbulence varied very little from 13.2-14.2% in one wake passing period, in comparison to the unsteadiness in the centre of the stator interaction region at the hub of 16.8-20.4%. Another periodic variation in the turbulence structure can be observed near region C located to the left of the shroud leakage flow. This region has maximum turbulence intensity of 13.2% at time $t/\tau=0.375$ and minimum turbulence intensity 9.6% at time $t/\tau=0.875$. This may be due to the interaction of the passage vortex near the stator casing. The rotor exit flow field is further investigated with the help of yaw and pitch angle data.

Figure 6.26 presents the unsteady variation of the contours of the relative yaw angle at rotor exit (plane 3) in one stator wake-passing period. The time-mean area integrated yaw angle at this location is -69.96 degrees. With respect to this angle, the flow overturning can be observed near the casing relating to the shroud leakage flow, near the hub corresponding to the hub secondary flow. The flow overturning was also observed at 30% blade height (near region A) to the right of the suction surface. The flow structure corresponding to the region A varied periodically with time and has a minimum angle of -75.25 degrees at $t/\tau=0.375$ and a maximum angle of -80.5 degrees at $t/\tau=0.875$. This overturned flow region also has maximum turbulence intensities (as observed in figure 6.25) at all the instants in time. The shroud leakage flow angles did not vary significantly over the wake-passing period, further confirming the negligible effect of the unsteadiness on the shroud leakage flow. In addition, the overturning region can be observed very close to the hub due to the rotor secondary flow.

The contours of the pitch angle of the flow can be used for identifying the secondary flow vortices in conjunction with yaw angle data. Figure 6.27 presents the unsteady variation of the pitch angle at the exit of the rotor in one stator wake-passing period. The flow coming out of the hub is considered positive and the flow going into the hub is negative pitch angle. Defining the secondary velocity vector in a three dimensional flow like this at the exit of the rotor is very difficult when spanwise variations of the flow angle is high. An alternate approach of Binder and Romey (1983) is used to identify vortices. The vortex can be identified in a contour plot of yaw angle as a region of contour lines parallel to the circumferential direction (the forced vortex core) and two sets of enclosed contours with their centres separated radially (free vortex core). A similar pattern rotated by 90^0 would be seen in the contours of pitch angle. Using this technique, the regions of rotor hub passage vortex, and a vortex rotating opposite to this near the hub (region A) is identified in figure 6.27. This opposite rotating vortex is stronger at $t/\tau=0.875$ and weak at $t/\tau=0.500$ and is periodic in one stator wake-passing period. The contours of the pitch angle together with the turbulence intensity data and yaw angle data indicate that this region is due to the stator passage vortex. The region corresponding to the rotor hub passage vortex can be seen at all instants of time and can be observed to be weak at time $t/\tau=0.500$ and strong at $t/\tau=0.875$.

The consequence of the vortex transport corresponding to the stator passage in the rotor blade on loss has already been discussed in section 6.5.1 and given in table 6.2. In addition to the steady numerical simulations discussed in this chapter, unsteady three-dimensional numerical simulations were also carried out for this test configuration to further understand various interactions taking place. These results will be discussed in chapter 7 in conjunction with other numerical and analytical models describing the vortex transport in the downstream blade row.

In addition to the vortex interaction with the downstream blade discussed above further useful information was gained from the measurements. The turbulence parameters, which characterise the flow (e.g. turbulence kinetic energy, integral length scales, turbulence diffusion rates and Kolmogorov's microscales of turbulence), were evaluated in the regions of the blade wake, passage vortices and in the free-stream regions. Higher integral length scales and turbulence diffusion rates were found in the centres of passage vortices and blade wakes while two orders of magnitude reduction in diffusion rate was found in the free-stream region. Further details of the investigation and discussion on the turbulence parameters are given in Appendix A2.

6.6 Conclusions

This chapter demonstrated the use of delta wings for simulating the passage vortices in a turbine. In wind tunnel experiments, it has been shown that the total loss generated behind a delta wing was almost constant with downstream axial distance for a constant velocity flow. On the other hand, the total loss increased with downstream axial distance for the accelerating flow condition. The generation of additional loss was considered to be due to the stretching of the delta wing vortex. This is discussed further in the next chapter.

The half-delta wings were fixed to a rotating hub in front of the stator blade row to simulate an incoming upstream rotor passage vortex and tested in the low-speed rotating rig. The loss measurements at the exit of the stator blade showed a significant increase in stagnation pressure loss due to the delta wing vortex transport. Most of the increase in stagnation pressure loss was due to the increase in stator secondary flow. The comparison of the stagnation pressure loss at stator exit with the datum configuration showed that additional losses were generated from the interaction of the delta wing vortex with the stator blade row. The increase in loss was 25% of the datum stator loss, demonstrating the importance of this vortex interaction.

The rotor exit flow was also affected by the interaction between the enhanced stator passage vortex and the rotor blade row. Flow underturning near the hub and overturning towards the midspan was observed, contrary to the classical model of overturning near the hub and underturning towards the midspan. The spanwise distribution of the exit flow angle was similar to the observation of Sharma *et al.* (1988). This flow behaviour was not observed either in the radially stacked turbine or in the 3-D turbine without the delta wings upstream. It was also observed that the strength of the passage vortex at the stator hub for the delta wing test case was much higher than the datum 3-D turbine case and the radially stacked turbine case. This indicates that this phenomenon was a function of the incoming vortex strength at rotor inlet.

In addition to the vortex interaction with the downstream blade discussed above further useful information was gained from the measurements. All the Reynolds normal and shear stresses were evaluated behind the stator, which describes the turbulence production rates in the flow. These showed that the regions corresponding to passage vortices, blade wakes and leakage vortices were anisotropic in nature.

Modelling Vortex Transport through the High Pressure Steam Turbine

7

7.1 Introduction

The interaction of the delta wing vortex with the stator blade row resulted in additional loss generation inside the stator blade row as discussed in chapter 6. This chapter further investigates the flow field using unsteady numerical simulations for understanding the interaction of the delta wing vortex with the downstream blade row. The entropy generated inside the delta wing, stator and rotor blade rows were evaluated using unsteady numerical simulations and a comparison has been made with the steady numerical simulations. Finally, a simple analytical model is proposed for evaluating the additional losses generated from the vortex transport in the downstream blade row.

7.2 Delta wing vortex transport

Unsteady numerical simulations were carried out with a three-dimensional multi-blade solver ‘UNSTREST’. The details of the solver and the computational grid used in the investigation are given in section 3.8.2. The test configuration consisted of 21 delta wings, 36 and 42 stator and rotor blades respectively. In the computation, the rig is assumed to consist of 18 delta wings and 36 stator and rotor blades (a total of 1,136,520 nodes) to keep the ratio of the number of blades to 1:1 or 1:2. Although the assumption of integer blade count change the solidity and in turn the secondary flow development of the delta wing and rotor blades, it is assumed that this does not effect the transport behaviour of the delta wing vortex. The computation time per node per iteration on a DEC Alpha 533 MHz computer is 9.8 micro sec.

Owing to very fine grid in the endwall regions and near the blade surfaces, large number of iterations (200,000) were needed to obtain a periodic solution. The pressure, density and velocity at the delta wing exit and at stator and rotor exits were monitored for a periodic solution and to establish the number of time steps required in one delta wing passing period. The same grid as used for steady simulations was employed for these unsteady simulations. The numerical investigations were carried out at the design condition of the turbine (i.e. flow coefficient of 0.351).

The numerical results are discussed by analysing the entropy function contours in blade-to-blade and quasi-orthogonal planes at various time instants over one vortex passing period. The entropy function is defined as exp(-Δs/R). For a cascade with uniform inlet flow, this function reduces to the stagnation pressure recovery coefficient (P_{02}/P_{01}). The reference stagnation pressure is taken as the mid-passage inlet stagnation pressure at the blade mid-span location. A value of 1.0 for the entropy function corresponds to the flow with no losses and any value less than 1.0 represents a loss. Figure 7.1 presents the contours of entropy function in the blade-blade plane, at 14.4% blade span. The selected plane passes through the middle of the vortex at the delta wing exit. The plot shows one delta wing blade passage and two stator and rotor blade passages.

The vortex generated from the delta wing is identified by a low entropy function region in figure 7.1(a) at time $t/\tau=0.0$. The flow in the second passage of the stator blade row is used for tracking the delta wing vortex through the stator. At time $t/\tau=0.0$, the delta wing vortex can be observed near the leading edge of the middle stator, having reached to the pressure side of the preceding stator in the upper passage. After 20% of the vortex passing period at time $t/\tau=0.2$, the vortex convects just beyond the leading edge of the middle stator blade. The vortex bends around the leading edge of the stator. The incoming vortex has entered the lower stator blade passage by the time $t/\tau=0.4$. The delta wing vortex is chopped by the downstream blade row in a similar way to the wake. The pitchwise variation in convection velocity across the stator passage is responsible for the distortion of the vortex centre-line. The bowing of the vortex can be observed in figure 7.1(d) at time $t/\tau=0.6$, due to the higher convection rate in the mid-passage flow compared to the blade surface flows. The bowed vortex tube appears to have two counter-rotating legs extending back to the leading edges of the adjacent blades. These are termed as the suction side leg and pressure side leg. The re-orientation or shearing of the vortex can be seen in figure 7.1(e) and 7.1(f). Reorientation and shearing of the vortex occurs because a fluid particle will convect along the suction surface at a much higher rate than the pressure surface. This leads to the stretching of the vortex on the pressure side leg. The delta wing vortex migrates radially also near the blade surfaces due to the dynamics of the image vortices. This is discussed later in the section.

The vortex stretching continues at time t/τ=1.0. By the time t/τ=0.2 (figure 7.1(b), passage 2), the suction and pressure legs are not together. This may be due to the radial upward migration of the pressure leg of the delta wing vortex. This is further discussed by investigating the flow at various axial locations in the stator row in the quasi-orthogonal plane. The two legs of the vortex after this location observed to convect downstream separately on the pressure and suction sides of the blade with the respective local flow velocities. The suction leg of the vortex can be observed to convect down in figs. 7.1(c) and 7.1(d). At the exit of the stator blade row, the blade wake can be observed as a combination of the surface boundary layers and the vortical fluid from the delta wing vortex. This stator blade wake enters the downstream rotor. The transport of the combined flow structure is observed to be similar to the stator wake transport inside the rotor blade as discussed in section 4.3. The stator wake undergoes bowing while entering the rotor blade row, stretching, and shearing while convecting through the blade row. The stator wake can be observed as a concentrated fluid on the suction surface with a tail going to the pressure side of the blade at rotor exit.

7.2.1 Unsteady Flow in the Stator Blade row- 'Delta wing' configuration

The interaction of the delta wing vortex with the stator blade is discussed using entropy function contours in four quasi-orthogonal planes as shown in figure 7.2. The axial locations of the quasi-orthogonal planes are located at 13% stator C_x distance upstream of the stator, and 31%, 61% and 82% stator C_x distance inside the upper and lower stator blade passages respectively. Only the contours between 0.998 and 0.98 were plotted for better visibility of all the axial sections simultaneously.

The instantaneous contours of the entropy function over one delta wing vortex passing period are presented in figure 7.2. The delta wing vortex can be identified in the first axial station as a loss core entering the stator computational domain. The delta wing vortex is moved towards the stator suction surface in both figures 7.2(b) and in figure 7.2(c). At time t/τ=0.6 (figure 7.2(d)), the incoming delta wing vortex can be observed at the second axial plane. After an additional 20% of the delta wing passing period (t/τ=0.8), a local increase in loss can be observed on the suction side of the stator blade in figure 7.2(e). It can be observed that the suction and pressure legs of the vortex are not connected together. The disappearance of the connection between the suction side leg and the pressure side leg might be occurring between the second and third axial planes. The suction side leg of the delta wing vortex can be observed as a bulge in the loss core near the suction surface, reaching the fourth axial plane by time t/τ=0.9 in figure 7.2(f). Another observation can be made regarding the convection rates of the delta wing vortex inside the stator row. The delta wing vortex took almost 60% of the vortex-passing period to convect from plane 1 to plane 2, which are separated by a distance of 44% stator C_x. While it took only 30% of the vortex-passing period to convect from plane 2 to plane 5, which are separated by a distance of 52% stator C_x. This indicates that after reaching the stator suction surface, the delta wing vortex convected at much higher velocity. This also indicates the amount of vortex stretching possible near the stator suction surface.

The delta wing vortex transport shown in figure 7.2 is also illustrated in figures 7.3 to 7.5 over one vortex passing period in a quasi-orthogonal plane. Figure 7.3 presents the contours of entropy function at 13% stator C_x distance upstream of the stator leading edge over one vortex passing period. The delta vortex can be identified near the pressure surface side of the stator in figure 7.3(a) at time t/τ=0.0 and can be tracked through the two stator passages over one delta wing vortex passing period. The radial location of the vortex is around 20% blade span.

Figure 7.4 presents the entropy function contours at 61% stator C_x distance inside the stator blade passage over one vortex-passing period. Loss is observed on both the suction and pressure surfaces of the stator due to the blade surface boundary layers. The loss region corresponding to the stator secondary flow can also be observed near the suction surface of the blade. There is a significant variation in the size and magnitude of the secondary loss core over one vortex passing period. The presence of a bulge in the loss region corresponding to the delta wing vortex is observed at time t/τ=0.3 in the first stator passage. The same vortex can also be observed at time t/τ=0.8 in the second stator passage. The pressure leg of the delta wing vortex can not be observed separately at any of the time instants. This observation in conjunction with the observations from figure 7.1 and 7.2 suggest that larger stretching of the vortex near the pressure surface resulted in the dissipation of the same through viscous mechanism. Hence, the pressure leg of the vortex can not be identified in any of the axial planes after 61% stator C_x inside the stator.

Figure 7.5 presents the entropy function contours at 8.4% stator C_x distance downstream of the stator trailing edge over one vortex passing period in a quasi-orthogonal plane. This plane is at the same location as that of the measurement plane for ‘rotating hub’ test case and the ‘delta wing’ test case as discussed in sections 5.9 and 6.4 respectively. By this axial plane, the delta wing vortex can no longer be distinguished from the rest of the flow. The secondary flow near the stator hub is significant and covers up to 50% of the passage width. A variation in the magnitude of the hub secondary flow can be observed with a maximum size of the secondary flow at time $t/\tau=1.0$. The delta wing vortex is transported with the stator secondary flow and cannot be observed separately. This concurs with the measurements carried out at the same location with a three-axis hot-wire shown in figure 6.15. This may be the reason why the turbulence intensity contours in figure 6.15 look to be steady.

A comparison of the contours of the measured turbulence intensity in figure 6.15 and the present unsteady predictions indicate the similar stator hub secondary flow structure. The stator hub passage vortex can not be observed as loss core in the middle of the blade passage in the numerical simulations, instead, it is attached to the stator blade wake. This discrepancy initially considered to be due to the numerical viscosity (SF=0.0036) present in the solver. Another calculation with much lower numerical smoothing (SF=0.0009) also indicated the similar stator hub secondary flow structure. This indicates that the discrepancy between the turbulence intensity measurements and the predicted entropy function contours may be due to the type of mixing model used in the present calculation. Nevertheless, the unsteady numerical simulations were in good agreement in predicting the location and size of the stator passage vortex at the hub.

7.2.2 Unsteady Flow in the Rotor Blade row- ‘Delta wing’ configuration

This stator viscous flow varying with time generates further unsteadiness while convecting through the downstream rotor. The unsteady flow field in the rotor is discussed by studying the flow at three axial locations. The results are again presented as plots of entropy function contours in figures 7.6-7.8. The present ‘delta wing’ numerical test configuration has 18 delta wings, in comparison with 36 stator and rotor blades each. Hence, one appearance of the delta wing vortex behind the rotor can be expected over two-stator blade passing periods. No variation between the two subsequent rotor passages is observed in the unsteady measurements and unsteady simulations at the exit of the rotor. Hence, entropy function contours over only one stator-passing period are presented.

Figure 7.6 shows the results at 10% rotor axial chord upstream of the rotor leading edge over one stator blade-passing period. The stator blade wakes can be identified by the contours of lower entropy function in the middle of the passage. In addition to the stator wakes, the loss cores can be observed corresponding to the stator hub and casing passage vortices in figure 7.6(a). After 20% of the stator passing period at time $t/\tau=0.2$, the stator blade wake thickness reduced and the secondary flow at the hub is increased. At time $t/\tau=0.4$, the secondary flow near the hub has increased and also at the casing. At this instant in time (figure 7.6(c)), the wake is restricted to the middle of the blade for about 15% blade span, while the secondary flow at the hub and the casing covered rest of the blade span. The hub secondary flow can be observed for about 75-80% blade pitch. At time $t/\tau=0.6$, the stator wake and secondary flow fully occupies the blade height and the wake is no longer distinguishable from the stator secondary flow. After another 20% of vortex passing period at $t/\tau=0.8$, the magnitudes of the entropy function corresponding to the stator flow has reduced to lower levels. The stator exit flow is similar to the time $t/\tau=0.0$ after another 20% of the period at $t/\tau=1.0$. At this instant, again both the blade wake and the secondary flow are distinguishable.

It can be observed in figure 7.5 that the stator viscous flow variation with the rotor passing period is not substantial. The variation in the stator flow only restricted to the hub secondary flow. The time varying flow field presented in figure 7.6 shows otherwise. A large variation in the incoming stator flow to the rotor row. The axial plane is very close to the rotor leading edge (10% C_x upstream of the rotor leading edge). This suggests that this variation in the stator flow may be due to the potential field of the rotor. It can also be due to the passage vortex interaction of the stator with the rotor leading edge, which results in the turbulence generation as suggested by Binder *et al.* (1985). This phenomenon can be further investigated either numerically or by measurements.

The interaction between the significantly varying stator flow in time with the rotor blade flow is illustrated in figure 7.7. Figure 7.7 presents the entropy function contours at 50% rotor C_x distance inside the rotor blade passage from the leading edge over one stator-passing period. The loss core corresponding to the rotor secondary flow at the casing can be observed near the suction surface in both passages 1 and 2 at time $t/\tau=0.0$. After 20% of the stator passing period (figure 7.7(b)), the rotor secondary flow has increased at the hub, while the secondary flow at casing did not change much. A further increase in the rotor secondary flow can be observed at time $t/\tau=0.4$ near the hub and the casing. After an additional 20% of stator passing period (figure 7.7(d)), loss regions (region 1) can be observed in both the rotor passages at about 25% blade span. The increase in casing secondary flow is also visible at this time instant. By observing the region 1 at all the instances in time, it can be concluded that this region is periodic in one stator wake passing. This indicates that this region is due to the stator flow. This region can be observed to move towards the pressure surface by time $t/\tau=1.0$. This is due to the differential velocity of the stator flow near the suction and the pressure surface. The stator flow initially can be observed near the suction surface due to higher convection rates than the pressure surface. At time $t/\tau=0.2$, this stator viscous flow cannot be observed while at time $t/\tau=0.6$, the stator flow is just passing through the present axial plane. At these corresponding times, the rotor secondary flow at the hub is maximum at time $t/\tau=0.2$ and minimum at time $t/\tau=0.6$. This indicates that the presence of the stator flow at this axial plane reduced the development of the rotor secondary flow near the hub.

Further downstream, the rotor exit flow field at 10% rotor C_x distance downstream of the rotor trailing edge is presented in figure 7.8 over one stator wake passing period. This axial plane is at the same location as that of the measurement plane for 'rotating hub' test case and the 'delta wing' test case as discussed in sections 5.9.2 and 6.5.2 respectively. The loss generating regions are identified corresponding to the rotor blade wake and the hub and casing passage vortices. The contours of the entropy function show that the secondary flow at rotor hub varied significantly over one stator-passing period. The presence of the stator hub passage vortex near the suction surface can be observed from $t/\tau=0.4$ to $t/\tau=0.6$ and on the pressure surface at $t/\tau=0.8$. A comparison of the contours of the measured turbulence intensity in figure 6.25 and the present unsteady predictions indicate the similar rotor hub secondary flow structure. The rotor hub passage vortex in the predictions is much larger than the turbulence intensity measurements indicating this may be due to the lower blade solidity modelled than the test conditions. The rotor casing shroud leakage flow can not be predicted by the unsteady simulations, as it is not modelled in the calculation. The radial upward migration of the stator secondary flow can be observed at this plane, which is in agreement with the hotwire measurements.

The unsteady numerical predictions combined with the hotwire measurements presented in section 6.4.2 indicate that the delta wing vortex transport inside the stator was different to the earlier models presented for the radially stacked turbine in section 4.5 and for the 3-D turbine in section 5.9.2. The stretching of the delta wing vortex near the pressure surface resulted in disconnecting the pressure side leg of the vortex from the suction side leg. The two legs of the delta wing vortex later convected downstream with their respective surface boundary layers. The effect of the viscosity in the flow on the observed vortex transport in the stator blade row is further investigated and the results are discussed in the following section.

7.3 Inviscid transport

The effect of viscosity on the transport of the delta vortex is investigated by switching off the viscous terms in the downstream blade rows (stator and rotor) after generating the vortex in the delta wing row. The only dissipative mechanisms in the flow field are the artificial and numerical viscosities present in the numerical scheme. Unsteady numerical simulations were carried out for this special case. The results are summarised in figure 7.9. The contours of the entropy function in a blade-blade plane at 14.4% blade span is presented in figure 7.9(a). The flow-fields at four axial locations inside the blade are shown in figures 7.9(b)-(e). The axial locations of the quasi-orthogonal flow fields were kept identical to the viscous calculation presented in section 7.2. Four quasi-orthogonal views of the flow field in the stator blade row are presented at 13% C_x upstream of the stator blade row, 22% and 61% C_x inside the stator row and 8.4% C_x downstream of the stator.

The delta wing vortex can be identified with contours of lower entropy function as shown in figure 7.9(a). After convecting into the downstream stator, the vortex travels inside the blade row with the mainstream flow. The vortex, which is about to enter the stator blade row, can be observed in figure 7.9(b). Figure 7.9(c) presents the flow field at 22% C_x inside the stator. The delta vortex can be observed on the pressure side of the blade. The radial migration of the vortex flow is observed on both the suction and pressure sides of the blade row. Another 40% inside the blade row at 61% C_x, the delta wing vortex cannot be identified separately as shown in figure 7.9(d). This suggests that between 22% C_x and 61% C_x, the suction leg of the vortex tube is separated from the pressure leg. Figure 7.9(e) presents the data at 7% C_x downstream of the stator. A large secondary flow corresponding to the stator blade row can be observed near the hub. The unsteady flow-field at this plane over one vortex passing period revealed the variation in secondary flow similar to the one shown in figure 7.5. From this analysis, it can be said that the vortex transport in this investigation is similar to that discussed in section 7.2. This indicates that viscosity of the flow has no significant effect on the vortex transport phenomena. The effect of the interaction between the vortex and the downstream blade on the unsteady loss generation is discussed in the following section using numerical simulations of the flow.

7.4 Unsteady Loss and Entropy Production

The steady and unsteady predicted flow field was investigated to determine the contribution of the unsteady flow to the stage loss. The steady and unsteady numerical simulations have been carried out with identical grids and solved using identical numerical schemes, mixing lengths, relaxation parameters, and boundary conditions. Figure 7.10 presents the variations of the mass flow averaged entropy function with meridional distance for the steady and the unsteady calculations. The variations of the entropy function for unsteady simulations were plotted at three time instants over a delta wing passing period. The locations of the interface planes, the leading and the trailing edges of the blade rows were marked. At any meridional location, the deviation of the entropy function from the value of 1.0 gives the cumulative loss generated up to that location.

It can be observed in figure 7.10 that up to the first 72mm of the meridional distance there is little variation in entropy function values between the steady and unsteady simulations and the magnitude is very close to 1.0. This indicates little or no additional loss generation until the leading edge of the delta wing. From the leading edge of the delta wing, the values of the entropy function start to reduce representing the loss generation from the development of the delta wing vortex. The reduction in entropy function continues until the delta wing row interface plane at 92mm. After the trailing edge of the delta wing, the loss generation is from the mixing of the delta wing vortex with the surrounding flow. At the delta wing row interface plane, the sudden decrease in the entropy function value for steady calculations can be observed due to the instantaneous mixing process being employed. For the unsteady calculations, the entropy function continues to reduce with meridional distance without any steep variations across the interface plane. The difference in entropy function between the two simulations at any axial location after the delta wing interface plane can be considered due to the loss generated from unsteady flow.

It can be observed that after the delta wing interface plane (figure 7.10) until the stator mid-chord location, the value of the entropy function for unsteady simulations is higher than the steady simulations indicating lower losses from the unsteady calculations. The higher losses from the steady calculations is due to the instantaneous mixing of the delta wing vortex in the interface plane rather than mixing in the downstream blade row. Towards the end of the stator blade, the entropy function for the unsteady simulations is lower than the steady case indicating higher losses. The increase in loss is due to the interaction of the delta wing vortex with the stator blade. The difference between the steady and time average unsteady computations at the stator interface plane in terms of efficiency is 0.23 % in 2.43% loss in efficiency until that location. A loss audit at 8.4% stator C_x downstream of the stator trailing edge can be carried out using the results from steady and unsteady simulations. This axial location is the same as the measurement location and hence an easy comparison can be made with the measurements. The loss from the unsteady interaction can be defined as

$$\omega_{unsteady} = (\omega_{DW+stator} - \omega_{stator}) - \omega_{DW} \tag{7.1}$$

where ω is the loss in efficiency from the inlet to the stage. The total loss in efficiency upto the 8.4% stator C_x ($\omega_{DW+stator}$) can be evaluated from unsteady simulation. The loss in efficiency from the stator (ω_{stator}) only and from the delta wings (ω_{DW}) only can be calculated from steady simulations. The unsteady loss can then be calculated as 0.53% in efficiency. This loss also includes the mixing loss of the delta wing vortex. The unsteady loss as a percentage of the stator loss is 22%, which is in good agreement with the measured value of 25% of the stator loss coefficient (Y) as shown in section 6.4.1.

Similarly, large differences between the two simulations in the rotor blade row can be observed from stator interface plane (226mm) to stage exit (367mm) of the meridional distance. The predicted efficiencies for the turbine stage with steady and time average unsteady simulations are 95.13% and 94.68% respectively. This indicates the reduction in efficiency from unsteady flow as 0.45 %.

Though the overall contribution of the unsteady flow to stage loss can be evaluated based on figure 7.10, it is difficult to quantify the additional loss generated due to the interaction of the delta wing vortex with the downstream stator. For this, the entropy generation rate in each cell is calculated by evaluating the entropy fluxes into and out of the control volume similar to that discussed in section 5.8. The analysis is carried out for each cell in the computational domain with unsteady numerical simulations.

The entropy production rate (J/sec. K) is calculated for every time step in a time accurate unsteady simulation and integrated over one delta vortex-passing period. The results are presented at three axial locations in the stator blade row in figures 7.11(a)-(c). The corresponding contours of the entropy function are also presented on the top of entropy production rate plots. Figure 7.11(a) present the entropy generation rate contours at 10% stator C_x distance behind the delta wing row. The delta wing vortex and the corresponding entropy production rate can be identified in the figure. The entropy production rate at 20% C_x inside the stator is presented in figure 7.11(b). The delta wing vortex is evident inside first stator passage in the entropy function plot. This plot presents the instantaneous flow field, unlike the entropy production rate contours which is, integrated over one delta wing passing period. After a further 30% stator C_x distance, the flow inside the stator at 50% C_x is shown in figure 7.11(c). By the time the vortex tube reached this location, it is stretched on the pressure surface that resulted in the suction and the pressure legs of the vortex being separated. The entropy production rate on the suction side of the blade is significant compared to the pressure side because of the higher flow velocity on the suction side. Similar analysis can be carried out inside the rotor blade to understand the interaction of the stator flow on the rotor row.

In summary, the unsteady numerical simulations were used for understanding the vortex transport in the downstream blade row. The results were further analysed for identifying the entropy producing regions in the unsteady flow field. It has been shown that the additional loss was generated from the interaction of the vortex with the downstream flow. An attempt has been made to model the vortex transport inside the downstream blade and this is discussed in the following section.

7.5 Analytical Model

The previous sections of this chapter indicated that the vortex transport in the downstream blade row is complex. It has also shown that the vortices undergo bowing, shearing and stretching as they pass through the downstream blade row. The vortex transport also generates additional losses in the process. It is helpful to use unsteady Navier-Stokes numerical simulations to calculate the contribution of unsteady losses from vortex interactions. Nevertheless, designers would like to know the global effects of the vortex interaction without resorting to full unsteady Navier-Stokes simulations. Hence, an attempt has been made to simplify various aspects of the vortex transport to find a simple analytical solution for the calculation of unsteady loss at the preliminary design stage.

One of the first studies of secondary flow generation using analytical modelling was proposed by Hawthorne (1955). He modelled the incoming boundary layer as a vortex filament in the tangential direction and showed that as this vortex filament convected through the downstream blade row, it produced secondary, trailing filament and trailing shed vorticity. Smith (1955) presented a theory for the determination of secondary vorticity by considering the disposition of a vortex line as it moves through the blade passage with fluid particles. This model can be used for an inclined vortex filament unlike that of Hawthorne (1955). A comprehensive review on the generation of secondary vorticity in cascades and blades is given by Lakshminarayana and Horlock (1963) and by Hawthorne (1967). In the present investigation, a model similar to that used by Smith (1955) was used and analysis can be carried out for the transport of the vortex in the downstream blade row.

Figure 7.12 illustrates the kinematic model used in the present investigation. The incoming passage vortex (AP) can be considered as a concentrated vortex filament, which is at an angle to the main flow direction. The incoming freestream flow has a velocity of W_1 with inlet flow angle of θ_1. The exit flow has a velocity of W_2 with flow angle θ_2. The incoming vortex filament enters the downstream blade row inclined at an angle of γ_1 to the tangential direction as shown in figure 7.12. The assumptions used in the present analysis are

(a) The flow is incompressible and inviscid
(b) These is one vortex tube per passage at any instant in time
(c) The mass flow associated with the trailing filament vorticity is small and can be neglected. There is no variation in blade loading in the spanwise direction resulting in negligible trailing shed vorticity.
(d) The vortex tube diameter is small compared to the blade chord. It will be represented as a straight line at exit. The vortical flow is convected by the mean flow of the passage.

As the vortex filament is inclined to the flow direction, it first encounters the blade leading edge near the pressure surface while convecting downstream. After travelling distance ‘e’ with the freestream velocity W_1, the vortex filament now reaches the suction surface of the succeeding blade row. The vortex filament moves with the fluid downstream and satisfies the theorems of Kelvin and Helmholtz. The vortex will always be composed of the same particles and its motion may be followed until it is downstream of the blade row. Owing to the different velocities on the concave and convex surfaces of the blade, it will take up different position at the exit of the row (DQ). The length and the orientation of the vortex segment leaving the blade row can then be determined from the difference between the times at which the pressure and suction side legs of the vortex are discharged.

In the time taken by the particle on the pressure surface from leading edge at point ‘A’ to trailing edge point ‘D’, the particle near the suction surface would have travelled to point ‘Q’ which is at a distance of (f - (eW_2/W_1)) away from the blade trailing edge. This is due to the higher flow velocities on the suction surface compared to the velocity on the pressure surface. The time taken by the flow at point ‘P’ to convect to the blade leading edge at ‘B’ is given as e/W_1. When expressed with respect to exit velocity, the flow at point ‘P’ would have moved by eW_2/W_1 at the blade exit. The time ‘f’ represents the difference in transit time of a particle travelling from the leading to trailing edge of the blade when passing along the suction surface and when passing along the pressure surface. This is given as follows,

$$f = W_2 \, . \, \Delta t = W_2 \, . \int \frac{ds}{W_s} \tag{7.2}$$

The integral $\int ds / W_s$ is taken around the surface of the blade. The integral is approximated by an expression given by Smith (1955) as

$$\int \frac{ds}{W_s} \approx \frac{\Gamma}{W_\infty{}^2} \tag{7.3}$$

Where W_∞ is the vector mean of the inlet and exit velocities and Γ is the blade circulation. The blade circulation is given by

$$\Gamma = \frac{2\pi}{N}(r_1 W_{\theta 1} - r_2 W_{\theta 2}) \tag{7.4}$$

If the inlet and exit radius of the blade section is the same, the equation 7.4 becomes,

$$\Gamma = s(W_{\theta 1} - W_{\theta 2}) = s(W_1 \operatorname{Sin}(\theta_1) - W_2 \operatorname{Sin}(\theta_2)) \tag{7.5}$$

where, θ_2 is the exit angle of the flow and γ_2 is the exit angle of the incident vortex filaments as defined in figure 7.12. The vector mean of the inlet and exit velocities can be given as

$$W_\infty{}^2 = \left\{\frac{W_{x1} + W_{x2}}{2}\right\}^2 + \left\{\frac{W_{\theta 1} + W_{\theta 2}}{2}\right\}^2 \qquad (7.6)$$

From the conservation of mass, the ratio of outlet to inlet velocity is given as

$$\frac{W_2}{W_1} = \frac{\mathrm{Cos}(\theta_1)}{\mathrm{Cos}(\theta_2)} \qquad (7.7)$$

Substituting equations 7.5 to 7.7 in equation 7.2 and simplifying results in

$$f = \frac{4s\ (\mathrm{Tan}(\theta_1) + \mathrm{Tan}(\theta_2))}{\mathrm{Cos}(\theta_2)\ \left\{4 + \{\mathrm{Tan}(\theta_1) - \mathrm{Tan}(\theta_2)\}^2\right\}} \qquad (7.8)$$

Using the trigonometry and sine rule for the inlet and exit triangles shown in figure 7.12, the length of the inlet and exit vortex filament are given in terms of the blade geometry, flow angles and wake inlet angles as;

$$AP = \frac{s\,\mathrm{Cos}(\theta_1)}{\mathrm{Cos}(\theta_1 + \gamma_1)} \qquad (7.9)$$

$$DQ = \frac{s\,\mathrm{Cos}(\theta_2)}{\mathrm{Cos}(\gamma_2 + \theta_2)} \qquad (7.10)$$

The ratio of the exit vortex filament length to the inlet filament length can be defined as stretching ratio 'S_r' and can be calculated from the inlet and exit angles as,

$$sr = \frac{DQ}{AP} = \frac{\mathrm{Cos}(\theta_2)}{\mathrm{Cos}(\theta_1)}\frac{\mathrm{Cos}(\gamma_1 + \theta_1)}{\mathrm{Cos}(\gamma_2 + \theta_2)} \qquad (7.11)$$

The exit angle of the vortex filament γ_2 is evaluated from the relation

$$CQ = f - PB\frac{W_2}{W_1} = \frac{s\,\mathrm{Sin}(\gamma_2)}{\mathrm{Cos}(\theta_2 + \gamma_2)} \qquad (7.12)$$

Substituting equations 7.7 and 7.8 in 7.12 and simplifying gives a relation between the vortex inclination angle at blade exit as a function of the inlet and exit flow angles and vortex inclination angle at blade inlet as

$$\frac{1}{(1/\mathrm{Tan}(\gamma_2)) - \mathrm{Tan}(\theta_2)} = \frac{4\ (\mathrm{Tan}(\theta_1) + \mathrm{Tan}(\theta_2))}{\left\{4 + \{\mathrm{Tan}(\theta_1) - \mathrm{Tan}(\theta_2)\}^2\right\}} - \frac{\mathrm{Sin}(\gamma_1)\,\mathrm{Cos}(\theta_1)}{\mathrm{Cos}(\gamma_1 + \theta_1)} \qquad (7.13)$$

For the given blade geometry (s), inlet and exit flow angles (θ_1, θ_2) and vortex filament incidence angle at inlet (γ_1), the vortex filament angle at the blade exit and the stretching ratio can be calculated from equations 7.13 and 7.11. The results of the filament exit angle and the stretching ratio of the vortex filament for the turbine under investigation is given in figure 7.13.

Figure 7.13(a) presents the variation of the exit vortex filament angle (γ_2) as a function of the vortex filament angle at the blade inlet (γ_1) for the present turbine from equation 7.13. The results of the unsteady numerical simulations performed for the 3-D datum turbine case and delta wing case were used to evaluate the stator passage vortex angle at the rotor exit. These data are also shown in the figure for comparing with the analytical model results. As the inlet vortex angle approached zero degrees (normal vortex filament), the exit filament angle increased. This is due to the reduction of the time lag between the pressure and the suction legs of the vortex while entering the blade row. A reasonably good agreement between the unsteady simulations and the analytical model has been demonstrated.

Figure 7.13(b) presents variation of the stretching ratio as a function of vortex filament angle at the inlet (γ_1) from equation 7.11. The lengths of the passage vortex filament at the rotor inlet and at the rotor exit were evaluated from the entropy function contours in a blade-blade plane from unsteady numerical simulations. The results for the two turbine test cases are presented in figure 7.13(b). There is a reasonably good agreement between the simulations and the analytical model increasing the confidence in the analytical model. The effect of vortex stretching on the additional loss generation is discussed below.

The vortex filament can be approximated as a Rankine's combined vortex consisting of a core region containing all the vorticity and a outer region containing the irrotational outer flow. Then the tangential velocity distribution across the vortex with radius is given as

$$
\begin{aligned}
V_\theta(r) &= \left\{\frac{\Gamma}{2\pi\delta}\,\frac{r}{\delta}\right\} && 0 < r < \delta \\
&= \left\{\frac{\Gamma}{2\pi r}\right\} && \delta < r < R
\end{aligned}
\qquad (7.14)
$$

Where δ and R corresponds to the rotational inner core radius and irrotational outer core radius of the vortex respectively. Now consider a finite sized vortex tube as shown in figure 7.14(a). The tube is stretched with tension forces from a length l_1 to a length

$$l_2 = S_r \,.\, l_1 \tag{7.15}$$

Mass is conserved, and in incompressible flow, the volume is also conserved. This gives rises to the relation

$$\pi r_1^{\,2}.l_1. = \pi r_2^{\,2}.l_2 = \pi r_2^{\,2}.S_r.l_1 \tag{7.16}$$

$$\text{and} \quad r_2^{\,2} = r_1^{\,2} / S_r \tag{7.17}$$

The total rotational kinetic energy for the vortex with length l_1 is given as

$$\text{Kinetic energy} = \frac{1}{2} m r^2 \Omega^2 = \frac{1}{2} m V_\theta^{\,2}$$

where V_θ is the tangential velocity inside the vortex and varies as per equation 7.14. The total rotational kinetic energy for this vortex tube can be calculated by integrating the velocity distribution over radius. It is given as

$$\text{Total rotational kinetic energy} = \left[\int_0^\delta \frac{\Gamma}{2\pi\delta} \frac{r}{\delta} dr + \int_\delta^R \frac{\Gamma}{2\pi r} dr \right]^2 \tag{7.18}$$

After evaluating the integral and assuming the outer irrotational core radius is five times the rotational core, it can be simplified to

$$\text{Total rotational kinetic energy} = \frac{17\Gamma}{60\pi^2 \delta_1} \tag{7.19}$$

The assumption of irrotational core radius is for guidance only. As we are working on the ratio of outlet to inlet secondary kinetic energy, this value will not effect the solution. Similarly, the total kinetic energy for the vortex tube after stretching can be calculated as $\frac{17\Gamma}{60\pi^2 \delta_2}$. The ratio of the total kinetic energy after and before stretching can be given as

$$\text{Kinetic energy ratio} = \frac{17\Gamma / 60\pi^2\delta_2}{17\Gamma / 60\pi^2\delta_1} = \frac{\delta_1}{\delta_2} = \sqrt{S_r} \tag{7.20}$$

This relation shows that the increase in stretching ratio increases the secondary kinetic energy at blade exit. It can be shown that this total kinetic energy ratio for the vortex tube rotating as a body varies as the vortex-stretching ratio 'S_r'. Figure 7.14(b) presents the variation of the outlet to inlet total kinetic energy ratio as a function of vortex stretching ratio. The results from unsteady numerical simulations for two test cases described are used for evaluating the kinetic energy ratio. This increase in the secondary kinetic energy result in additional losses while mixing with the freestream flow. Despite the complexity of the actual vortex transport and the assumptions made in modelling the flow analytically, the presented model demonstrated the salient features of the vortex interaction. They are the effect of inlet filament angle of the vortex tube on the vortex stretching ratio and the effect of stretching ratio on kinetic energy ratio variation.

7.6 Conclusions

The unsteady numerical simulations were used for understanding the delta wing vortex transport inside the downstream blade row. A good agreement between the measurements and the unsteady numerical predictions has been observed for the mechanism of vortex transport inside the stator row.

The unsteady numerical simulation results were further analysed to identify the entropy producing regions in the unsteady flow field. It has been shown from entropy function variations that the additional loss was generated from the interaction of the vortex with the downstream flow. It has also been shown that the contribution of unsteady flow to the stage loss is significant. An analytical model was developed using the concepts of the kinematic vortex transport inside the downstream blade and assuming the incoming vortex to be as a concentrated vortex filament. The model was accurate enough in calculating the vortex filament exit angle and the increase in the exit secondary kinetic energy from vortex stretching inside the blade for the given blade parameters and inlet vortex angle. Despite the complexity of the actual vortex transport, the analytical model brought out the significant effects of vortex interaction with the downstream blade row very well.

Concluding Remarks and Suggestions for Future work

8

8.1 Introduction

It was shown in section 2.7 that in a high-pressure axial turbine the interaction of streamwise vortices with the downstream blade row resulted in large variations in the downstream flow field, especially on the development of the secondary flow. A maximum of 40% variation in secondary flow with time was observed in the rotor passage over one stator passing period by Sharma *et al.* (1988). The mechanism of the transport of passage vortices through the downstream blade row is not well understood. Though some experiments have been conducted to visualise the vortex interaction (LaFleur *et al.* (1988), Van de Wall *et al.* (1995)), they were restricted to water tunnel experiments in two-dimensional cascades at very low Mach numbers and Reynolds numbers. The additional loss generation due to the interaction of the vortex with a downstream blade row has also not been quantified. Hence, the objectives for the present investigations were

(1) Identify the unsteady phenomena of significance in high-pressure turbines.

(2) Understand the interaction of the streamwise vortices with the downstream blade row.

(3) Quantify the effect of upstream streamwise vortices on the loss of the downstream blade row.

(4) Translate this knowledge into design recommendations.

These objectives were met by a comprehensive experimental testing and numerical simulation programme. The interaction of the stator passage vortices with the rotor blade was studied in two very different turbine stages. The first corresponded to a radially stacked high-pressure gas turbine stage, while the second corresponded to a three dimensionally stacked and swept high-pressure steam turbine stage. The interaction of the streamwise vortices on the downstream blade row has been investigated by shedding the known streamwise vortices into the downstream blade row. This chapter summarises the conclusions from the present investigation and makes suggestions for further research.

8.2 Comparison of radially stacked and 3-D stacked turbines

The three dimensionally stacked turbine was aimed at reducing secondary flows compared to conventional radially stacked designs. Although the two turbine stages have different design requirements, it is still possible to compare both turbines. Both turbines have the same blade heights, hub-tip ratio, blade numbers, mean radius, angular speed, stage reaction and nearly the same stage-loading coefficient. The two turbine configurations had different clearance configurations at the rotor tip. The three dimensional stage has a shrouded rotor while the radially stacked turbine had an unshrouded rotor.

A summary of the overall performance of both turbines is given in table 8.1. It is observed that the coefficient of stagnation pressure loss is reduced considerably in the 3-D stator compared to the radially stacked stator. The reduction in loss was due to the reduction in hub and casing secondary flows in the 3-D stator blade, which resulted from the combination of reduced endwall blade loading and aft loading of the blade near the endwalls. The coefficient of stagnation pressure loss in the rotor was also reduced considerably in the 3-D turbine compared to the radially stacked case. Most of the loss reduction came from reduced overtip leakage losses and reduced hub and casing secondary flows. The overtip leakage loss was the major contributor to the rotor loss in the radially stacked case. The efficiency of the 3-D stage was 2.78% higher than the radially stacked stage. The comparison between the two indicates the possible improvement that can be achieved through three-dimensional blading. The present 3-D blade design is not optimised, and there is the potential for further improving the performance as discussed in section 5.4.

8.3 Stator - rotor interaction and vortex transport

The interaction of the stator viscous flow features with the rotor blade row was investigated with the help of flow visualisation, and both steady and unsteady measurements and simulations.

The stator wake transport inside the rotor blade was investigated with the help of smoke flow visualisation. The bowing, stretching and shearing of the wake through the downstream rotor row were demonstrated in the flow visualisation. The net result of the above phenomena was that the wake appeared to be concentrated on the suction surface at the exit of the rotor passage with a tail stretching back to the leading edge of the pressure surface of the adjacent blade.

The stator hub passage vortex is chopped by the downstream blade row in a similar way to the wake. It is then convected through the rotor passage at the freestream velocity. The bowed vortex tube near the inlet to the rotor appeared to develop two counter-rotating legs extending back to the leading edges of the adjacent blades. These were termed the suction side leg and pressure side leg. The pitchwise variation in convection velocity across the passage is responsible for the bowing of the vortex tube at rotor inlet. The two legs of the incoming passage vortex then convects with the respective velocities on the blade surfaces. An analysis of the vortex dynamics has shown that the stator passage vortices move radially outwards near the pressure surface and move radially inward near the suction surface due to the effect of the image vortices inside the blade surfaces. In addition to the vortex dynamics, the static pressure gradient also effects the transport of the stator passage vortex through the rotor blade. The results are discussed for the radially stacked turbine and the 3-D turbine separately.

8.3.1 Radially stacked turbine

The static pressure gradient across the rotor blade passage resulted in the migration of the pressure side leg of the stator passage vortex to the rotor suction surface, similar to the endwall boundary layer. The presence of the rotor passage vortices affects the transport of the stator passage vortices through the rotor. The kinematic interaction between the stator and the rotor passage vortices has two effects. Firstly, the suction side leg of the stator passage vortex was displaced radially upward over the developing passage vortex of the rotor blade. Secondly, the pressure side leg of the stator passage vortex was entrained into the rotor passage vortex. The pressure side leg of the stator passage vortex had the same direction of rotation as that of the rotor passage vortex at the hub, which accelerated the entrainment of the pressure side leg into the rotor passage vortex.

The presence of stator viscous flow features in the rotor significantly enhanced the rotor secondary flow. It was also shown that the rotor secondary flow at the hub moved radially by 20% span, and circumferentially by one fourth of the rotor pitch, and varied in size and strength over one stator-passing period.

8.3.2 Three dimensionally stacked turbine

The transport of the stator passage vortices inside the rotor blade was slightly different to the radially stacked blades. The lower blade loading near the endwalls in conjunction with the higher blade loading in the blade mid-span resulted in the strong radial static pressure gradient towards the mid-span. In addition, the lower blade loading near the endwalls resulted in lower static pressure gradient across the rotor passage. The radial pressure gradient on the pressure surface combined with the lower static pressure gradient across the passage resulted in the pressure leg of the stator passage vortex to migrate to the higher radial location from the action of image vortices inside the walls. The suction leg of the stator passage vortex is entrained in the rotor passage vortex at the hub. This is due to the lower strength of the stator passage vortex resulting from the three-dimensionally stacked stator blades.

The presence of the stator secondary flow at the rotor exit reduced the rotor secondary flow in contrast to the radially stacked case. The strongest rotor secondary flow occurs in the absence of the stator flow. Also, the lower velocities on the pressure surface resulted in the lower mass flow being effected by the pressure leg of the stator passage vortex. All these effects translated to lower interaction loss compared to a radially stacked case. This indicates that the three dimensional blading performed better from the perspective of stator viscous flow interaction with the rotor blade.

8.4 Delta wing vortex - stator interaction

Previous studies concerning vortex interaction in the literature were conducted in a multistage environment where various forms of secondary flows occur simultaneously within the blade row. Difficulty is encountered in isolating a particular secondary flow, then understanding its cause and effect. The use of delta wings for simulating the passage vortices in a turbine has been demonstrated in the present investigation. This gives the flexibility of varying the shed vortex size and strength without redesigning the blade rows to simulate the same effect.

In wind tunnel experiments, it was shown that the total loss generated behind a delta wing was almost constant with downstream axial distance for a constant velocity flow. By contrast, the total loss increased with downstream axial distance for the accelerating flow condition. The generation of additional loss was considered to be due to the stretching of the delta wing vortex.

The half-delta wings were fixed to a rotating hub in front of the stator blade row to simulate an in coming upstream rotor passage vortex and tested in the low-speed rotating rig. The loss measurements at the exit of the stator blade showed a significant increase in stagnation pressure loss due to the delta wing vortex transport. Most of the increase in stagnation pressure loss was due to the increase in stator secondary flow. The comparison of the stagnation pressure loss at stator exit with the datum configuration showed that additional losses were generated from the interaction of the delta wing vortex with the stator blade row.

The delta wing vortex transport inside the stator blade row was different to the earlier models presented for the radially stacked turbine and the 3-D turbine. The differential velocities on the pressure and the suction surfaces of the stator, combined with the lower streamwise momentum of the delta wing vortex, resulted in the stretching of the vortex near the pressure surface. This stretching eventually dissipated the pressure side leg of the vortex and disconnected the pressure leg from the suction side leg. The two separated legs of the delta wing vortex later convected downstream with their respective surface boundary layers.

The rotor exit flow was also affected by the interaction between the enhanced stator passage vortex and the rotor blade row. Flow underturning near the hub and overturning towards the midspan was observed, contrary to the classical model of overturning near the hub and underturning towards the midspan. The spanwise distribution of the exit flow angle was similar to the observation of Sharma *et al.* (1988). This flow behaviour was not observed either in the radially stacked turbine or in the 3-D turbine. It was also observed for the delta wing test configuration, that the strength of the incoming stator passage vortex at the hub was much higher than the datum 3-D turbine case and the radially stacked turbine case. This indicates that this phenomenon was a function of the incoming vortex strength at the rotor inlet. The effect of stator and rotor blade viscosity on the vortex transport was found to be negligible in the investigation in which the viscous terms in the downstream of the delta wing row were switched off.

It was difficult to model all aspects of the vortex transport with analytical methods. Nevertheless, designers would like to know the global effects of the vortex interaction without resorting to full unsteady Navier-Stokes simulations. Hence, an analytical model was developed using the concepts of the kinematic vortex transport inside the downstream blade, and assuming the incoming vortex as a concentrated vortex filament. The model was accurate in calculating the vortex filament angle at the blade exit and the increase in the exit secondary kinetic energy from vortex stretching inside the blade for the given blade parameters and inlet vortex filament angle. Despite the complexity of the actual vortex transport and the assumptions made in modelling the flow, the analytical model captured the significant effects of vortex interaction with the downstream blade row.

In addition to the main conclusions discussed above further useful information was gained from the measurements and simulations. The turbulence parameters, which characterise the flow (e.g. turbulence kinetic energy, integral length scales, turbulence diffusion rates and Kolmogorov’s microscales of turbulence) were evaluated in the regions of the blade wake, passage vortices and in the free-stream regions. Higher integral length scales and turbulence diffusion rates were found in the centres of passage vortices and blade wakes, while two orders of magnitude reduction in diffusion rate was found in the free-stream region. In addition, all the Reynolds normal and shear stresses were evaluated, which in combination with the velocity gradients in the flow, describe the turbulence production rates in the flow. These showed that the regions corresponding to passage vortices, blade wakes and leakage vortices were anisotropic in nature. These Reynolds stresses and turbulence parameters were considered useful as test data in the development of advanced turbulence models.

8.5 Unsteady loss

It is known that unsteady performance measurements are complicated by the lack of sufficient frequency response in unsteady temperature measurement, which is essential in calculating the entropy of the fluid. Hence, the unsteady numerical simulations were used in evaluating the contribution of unsteady flow to the stage loss.

Various definitions are possible in defining unsteady loss. One such definition relies on the assumption that the entropy produced within the flow field will eventually pass through the exit boundary of the stage. By evaluating the entropy flux passing through the stage exit boundary one can calculate the stage loss. The unsteady loss is then defined as the difference between the average entropy flux passing through the exit boundary, during one stator passing cycle, of an unsteady calculation and the corresponding entropy flux from a mixing plane steady calculation. The unsteady loss was found to be 33% of the steady stage loss using the results of the CFD computations for the radially stacked turbine stage.

The stage loss was also calculated by evaluating the entropy generation in a control volume and summing such control volumes in a blade passage. It was then possible to calculate the entropy increase for the whole blade row. Using this concept, a method was developed to calculate the loss generation in the blade row from numerical simulations, identifying the contribution of various regions of the blade row towards the total blade loss. It was observed that the maximum percentage of the loss was generated near the suction surface of the blade.

Similar analysis was carried out using the results from the unsteady numerical simulations, where the entropy generation in each control volume was integrated over one stator passing period. The unsteady loss generation from both the stretching of the stator passage inside the rotor blade, and the compression of the mid-pitch stator wake near the rotor throat region, was shown for the ‘datum three dimensionally stacked’ turbine configuration.

In an another investigation, rotating delta wings were used to shed vortices into the stator blade row. The comparison of the measured stagnation pressure loss at stator exit with the datum configuration showed that additional losses were generated from the interaction of the delta wing vortex with the stator blade row. The vortex interaction increased the stator loss by 25% above the datum configuration.

The additional loss generation from the interaction of the delta wing vortex with the stator was also illustrated by evaluating the integrated entropy generation rate inside the stator row.

8.6 A note on the numerical simulations and on the measurements

The detailed measurements in the two turbine test cases, in conjunction with the steady and unsteady numerical simulations, have helped to draw some conclusions about the suitability of the numerical simulations. The steady numerical simulations gave a good prediction of the flow behind the stator blade (first blade row). They were not accurate in predicting the flow field behind the rotor. This was mainly due to the inability in predicting the interaction between the stator and the rotor flows. The unsteady numerical simulations were successful in predicting accurately the flow near these interaction regions. It would be ideal to use unsteady numerical simulation for design purposes. However, the large computation times required for calculating unsteady flows makes this not possible. Instead, the designer has to note the deficiencies of the steady simulations near the flow interaction regions and use unsteady simulations for an analysis of the final design.

The present unsteady numerical simulations are based on solving the Reynolds averaged Navier - Stokes equations. They model the turbulence with a simple mixing length model. These calculations are accurate to predict the primary flow variables such as pressure, velocities, flow angles and entropy. It has also been shown using measurements that the regions corresponding to passage vortices, blade wakes and leakage vortices were anisotropic in turbulence. Hence, simulations based on advanced turbulence models which assume the flow to be isotropic may also have similar disadvantages as the simple mixing length models in modelling the turbulence. An alternative is to use large eddy simulation models (LES), these require huge computing resources and time and are not yet in use as general-purpose tools. Hence, for the designer, it seems to be sufficient to use simple mixing length model based simulations for predicting the unsteady flow field.

It has to be noted that there are uncertainties in the measurements as well. The five-hole pneumatic probes, the single slanted hot-wire and the three axis hot-wire were calibrated in shear free steady flows. The regions corresponding to the vortices and blade wakes are highly sheared and unsteady giving rise to an uncertainty in the measurements. Another uncertainty in the measurements is that the loss investigation behind the rotor was carried out using a five-hole probe. The acquisition time was selected in such a way that it corresponds to ten rotor revolutions. This measurement is considered to give the time average performance behind the rotor.

8.7 Suggestions for future work

There are several areas of interest highlighted by the current work, which warrant further investigation. These are outlined below.

It has been observed in section 7.2 that the variation in the stator flow with time only restricted to the hub secondary flow. The time varying flow field presented at 10% C_x upstream of the rotor leading edge showed otherwise. A large variation in the incoming stator flow was observed. The axial plane is very close to the rotor leading edge suggesting that this variation in the stator flow may be due to the potential field of the rotor. It can also be due to the passage vortex interaction of the stator with the rotor leading edge, which results in the turbulence generation as suggested by Binder *et al.* (1985). This phenomenon can be further investigated either numerically or by measurements.

In the ‘delta wing’ test configuration, flow underturning near the hub and overturning towards the midspan was observed at rotor exit, contrary to the classical model of overturning near the hub and underturning towards the midspan. Further analysis indicated that this phenomenon is a function of the strength of the stator passage vortex. This can be further investigated by a parametric study of the incident vortex strength, incident vortex angles and vortex size and the stage losses. This investigation can be conducted both experimentally and numerically using delta wings.

As vortex interaction with the downstream blade row was shown to result in significant additional loss, it is suggested that a detailed investigation be carried out in a low speed environment. The vortex-passing rig can be considered similar to a bar-passing rig for understanding the vortex interaction with the blade row. This type of rig would permit visualising and measuring the unsteady flow field inside the blade row, which is very difficult in a low speed rotating turbine rig. This will also allow further understanding of the transport mechanism of the vortex in the downstream blade row.

Further experiments can be undertaken with increased free stream turbulence representing the test environment to evaluate the effect of free stream turbulence on the vortex interaction. This could be simulated by placing wire gauze in front of the stator blade row. The unsteady measurements in the present investigation were carried out with a three axis hot-wire probe. The measuring volume of the probe is around 2mm, which is large for more detailed investigation of Reynolds stresses, turbulent diffusion rates and integral length scales. The use of laser velocimetry can reduce the measurement volume and hence will be better suited for the more accurate measurement of the flow inside the turbine.

Turbine designers have the need to estimate the aerodynamic impact of the vortex interaction with the downstream blade row without always performing time consuming three-dimensional unsteady computer simulations. The analytical model developed in this thesis for modelling vortex transport through the downstream blade rows can be used to estimate the increase in secondary kinetic energy from this interaction. This model used in conjunction with the traditional secondary flow analytical prediction models has the potential of predicting the total loss generated in a turbine stage. This can be used as a design tool in the preliminary stages of the design system.

It has been observed in the present investigation that secondary flow modelling is strongly influenced by shroud leakage and hub bleed effects. The unsteady numerical simulations used in the present investigation do not model these secondary flows. Multi block computations can model these secondary flow paths, such as shroud leakage and hub bleed flows, and will improve the predictions with unsteady simulations.

8.8 Concluding Remarks

The present work has demonstrated that the vortex interaction with the downstream blade row can be significant in terms of additional loss generation and its effect on the flow field. The effect of vortex interaction in a three-dimensionally stacked turbine stage was less than the corresponding radially stacked turbine stage. The kinematic vortex transport models were presented for the radially stacked turbine and three-dimensionally designed turbine. The additional loss generation due to the vortex interaction with the blade was evaluated from both experiments and computations. An analytical model was developed using the concepts of the kinematic vortex transport inside the downstream blade. Such modelling can be very useful in the preliminary design stage.

References

Adamczyk, JJ, 1985, ‘Model Equation for Simulating Flows in Multistage Turbomachinery’, ASME Paper No. 85-GT-226.

Adamczyk, JJ, 1996, ‘Wake Mixing in Axial Flow Compressors’, ASME Paper No. 96-GT-029.

Adamczyk, JJ, 1999, ‘Aerodynamic Analysis of Multistage Turbomachinery Flows in Support of Aerodynamic Design’, IGTI Gas Turbine Scholar Lecture, 1999.

Arts, T, Boerrigter, H, Carbonaro, M, Charbonnier, JM, Degrez, G, Olivari, D, Riethmuller, ML and Van den Braembussche, RA, 1994, ‘Measurement Techniques in Fluid Dynamics’, VKI LS-1994-01, 1994.

Atkin, CJ and Squire, LC, 1992, ‘A Study of the Interaction of a Normal Shock Wave with a Turbulent Boundary Layer at Mach Numbers between 1.3 and 1.55’, Eur. Jnl. of B/Fluids, Vol. 11, No. 1.

Bearman, PW, 1971, ‘Corrections for the Effect of Ambient Temperature Drift on Hot-wire Measurements in Incompressible Flow’, DISA Information, No. 11, May 1971.

Binder, A and Romey, R, 1983, ‘Secondary Flow Effects and Mixing of the Wake behind a Turbine Stator’, ASME Jnl. of Engineering for power, Vol. 105, pp40-46, January 1983.

Binder, A, 1985, ‘Turbulent Production due to Secondary Vortex Cutting in a Turbine Rotor’, Jnl. of Engineering for Gas Turbines and Power, Vol. 107, October 1995, pp 1039-1046.

Binder, A, Forster, W, Mach, K and Rogge, H, 1986, ‘Unsteady Flow Interaction Caused by Stator Secondary Vortices in a Turbine Rotor’, ASME Paper No. 86-GT-302, 1986.

Bindon, JP, 1980, ‘Exit Plane and Suction Surface Flows in an Annular Cascade with a Skewed Inlet Boundary Layer’, Intl. Jnl. of Heat and Fluid Flow, Vol. 2, No. 2, p57.

Bindon, JP, 1989, ‘The Measurement and Formation of Tip Clearance Loss’, ASME Jnl. of Turbomachinery, Vol. 111, pp 257.

Boletis, E and Sieverding, CH, 1991, ‘Experimental Study of the Three Dimensional Flowfield in a Turbine Stator Preceded by a Full Stage’, ASME Jnl. of Turbomachinery, Vol. 113, pp 1.

Boletis, E, Sieverding, CH and van Hove, W, 1983, ‘Effect of Skewed Inlet Endwall Boundary Layer on the 3-D Flowfield in an Annular Cascade’, Viscous Effects in Turbomachinery, AGARD-CP-351.

Burd, SW and Simon, TW, 1999, ‘Turbulence Spectra and Length Scales Measured in Film Coolant Flows Emerging from Discrete Holes’, ASME Jnl. of Turbomachinery, Vol. 121, July 1999.

Camp, TR, 1995, ‘Aspects of the Off-design Performance of Axial Flow Compressors’, Ph.D. thesis, Cambridge university.

Champagne, FH, Sleicher, CA and Wehrman, OH, 1967, ‘Turbulence measurements with Inclined Hot-wires, part 1’, Journal of fluid Mechanics, Vol. 28, No. 1, pp153-175, 1967.

Chanaud, RC, 1965, ‘Observations of Oscillatory Motion in Certain Swirling Flows’, Jnl. of Fluid Mechanics, Vol. 21, Part 1, pp 111-127.

Dawes, WN, 1990a, ‘A comparison of Zero and One Equation Turbulence Models for Turbomachinery calculations’, ASME Paper No. 90-GT-303.

Dawes, WN, 1990b, ‘Towards Improved Throughflow Capability: The Use of Viscous Flow Solvers in a Multistage Environment’, ASME Paper No. 90-GT-18.

Dawes, WN, 1993, ‘Simulating Unsteady Turbomachinery Flows on Unstructured meshes which adapt both in time and space’, ASME Paper No. 93-GT-104, 1993.

Dawes, WN, 1994, ‘A Numerical Study of the Interaction of a Transonic Compressor Rotor Overtip Leakage Vortex with the Following Stator Blade Row’, ASME Paper No. 94-GT-156.

Denton, JD and Cumpsty, NA, 1987, ‘Loss Mechanisms in Turbomachines’, Turbomachinery – Efficiency Prediction and Improvement, I. Mech. E., Paper No. C260/87.

Denton, JD and Johnson, CG, 1976, ‘Tip Leakage Loss of Shrouded Turbine Blades’, CEGB Report No. R/M/N630.

Denton, JD and Singh, UK, 1979, ‘Time Marching Methods for Turbomachinery Flow Calculation’, VKI Lecture Series 1979-07.

Denton, JD, 1982, ‘An Improved Time Marching Method for Turbomachinery Flow Calculation’, ASME Paper No. 82-GT-239.

Denton, JD, 1990, ‘The Calculation of Three Dimensional Viscous Flow Through Multistage Turbomachines’, ASME Paper No. 90-GT-19.

Denton, JD, 1993, ‘Loss Mechanisms in Turbomachines’, IGTI Gas Turbine Scholar Lecture, ASME 93-GT-435.

Denton, JD, 1994, ' Designing in Three Dimensions', AGARD Lecture series on 'Turbomachinery Design using CFD', May to June 1994.

Denton, JD, 1996, ‘3D Unsteady Multistage Multi passage Calculation (UNSTREST) - User’s Manual’, Whittle Laboratory, University of Cambridge, May 1996.

Denton, JD, 1997, ‘Multistage Turbomachinery Flow Calculation Program (MULTIP81) - User’s Manual’, Whittle Laboratory, University of Cambridge, April 1997.

Denton, JD, 1999, ‘Multistage Turbomachinery Flow Calculation Program (MULTIP99) - User’s Manual’, Whittle Laboratory, University of Cambridge, April 1999.

Denton, JD, 2000, ‘Multistage Turbomachinery Flow Calculation Program (MULTALL) - User’s Manual’, Whittle Laboratory, University of Cambridge, April 2000.

Deregel, P and Tan, CS, 1996, ‘Impact of Rotor Wakes on Steady State Axial Compressor Performance’, ASME Paper No. 96-GT-253.

Dring, RP, Joslyn, HD, Hardin, LW and Wagner, JH, 1982, ‘Turbine Rotor-Stator Interaction’, ASME Paper No. 82-GT-3, 1982.

Dunham, J., 1996, ‘Aerodynamic Losses in Turbomachinery’, AGARD-CP-571, January 1996.

Fritsch, G and Giles, MB, 1992, ‘Second order Effects of Unsteadiness on the Performance of Turbomachines’, ASME Paper No. 92-GT-389.

Gallus, HE, Zeschky, J and Hah, C, 1994, ‘Endwall and Unsteady flow Phenomena in an Axial Turbine Stage’, ASME Paper No. 94-GT-143.

Giles, MB, 1987, ‘Calculation of Unsteady Wake/Rotor Interactions’, AIAA paper No. 87-0006, Presented at AIAA 25th Aerospace Sciences Meeting, Reno, Nevada, 1987.

Giles, MB, 1990, ‘Stator-Rotor Interaction in a Transonic Turbine’, AIAA Paper No. 88-3093, Jnl. of Propulsion and Power, Vol. 6, No. 5, Sept.-Oct. 1990.

Goto, A, 1991, ‘Three Dimensional Flow and Mixing in an Axial Flow Compressor with Different Rotor Tip Clearances’, ASME paper 91-GT-89, June 1991.

Gregory-smith, DG, 1984, ‘Secondary Flow Theory in Straight and Annular Cascades’ in Secondary Flows and Endwall Boundary Layers in Axial Turbomachines, VKI Lecture Series, 1984-05.

Hancock, PE and Bradshaw, P, 1983, ‘The Effect of Free-stream Turbulence on Turbulent Boundary Layers’, ASME Jnl. of Fluids Engineering, Vol. 105, pp 284-289, 1983.

Harrison, S, 1989, ‘The Influence of Blade Stacking on Turbine Losses’, Ph.D. thesis. Cambridge university.

Hawthorne, WR, ‘The Applicability of Secondary Flow Analysis to the Solution of Internal Flow Problems’, Fluid Mechanics of Internal Flow, Gino Sovran, ed., Elsevier Publ. Co., 1967.

Hawthorne, WR, 1955, ‘Rotational Flow Through Cascades’, Jnl. of Mechanics and Applied Maths., Vol. 8, 1955.

Herbert, GJ and Tiederman, WG, 1989, ‘Comparison of Steady and Unsteady Flows in a Turbine Stator Cascade’, ASME Paper No. 89-GT-79.

Heyes, FJG, Hodson, HP and Dailey, GM, 1992, ‘The Effect of Blade Tip Geometry on the Tip Leakage Flow in Axial Turbine Cascades’, ASME Jnl. of Turbomachinery, Vol. 114, p 643.

Hinze, JO, 1975, ‘Turbulence’, McGraw-Hill, New York.

Hobson, DE and Johnson, CG, 1990, ‘Two stage Model Turbine Performance Characteristics’, Powergen Report No. RD/M/1888/RR90.

Hodson, HP and Dawes, WN, 1996, 'On the Interaction of Measured Profile Losses in Unsteady Wake-Turbine Blade Interaction Studies', ASME 96-GT-494.

Hodson, HP, 1983, 'Unsteady Boundary Layers on Axial Flow Turbine Rotor Blades', Ph.D. Thesis, Cambridge University, Cambridge.

Hodson, HP, 1985a, 'Measurements of Wake Generated Unsteadiness in the Rotor passages of Axial Flow Turbines', ASME Jnl. of Engineering for Gas Turbines and Power, Vol. 107, April 1985.

Hodson, HP, 1985b, 'A Blade-to-Blade Prediction of a Wake Generated Unsteady Flow', ASME Jnl. of Engineering for Gas Turbines and Power, Vol. 107, pp 337-344.

Hodson, HP, 1998, 'Blade Row Interactions in Low Pressure Turbines' in '*Blade Row Interference Effects in Axial Turbomachinery Stages*', VKI Lecture Series, 1998-02.

Hodson, HP, Baneighbal, MR and Dailey, GM, 1993, 'Three-Dimensional Interactions in the Rotor of an Axial Turbine', AIAA 93-2255, AIAA/ASME/SAE/ASEE 24th Joint Propulsion Conference, Monterey, CA, June 1993.

Hourmouziadis, J and Hubner, N, 1985, '3-D Design of Turbine Airfoils', ASME Paper 85-GT-188.

Howell, RJ, 1999, 'Wake Separation Bubble Interactions in Low Reynolds Number Turbomachinery', Ph.D. Thesis, Cambridge University.

Hughes, WF and Gaylord, EW, 1964, 'Basic Equations of Engineering Science', Schaum's outline series, McGraw Hill.

Jennings, IK and Shin, CT, 1993, 'An Exhaustive Study of the Flow in a Transonic Rotor', 2nd International Symposium on Experimental and Computational Aerothermodynamics of Internal Flows, Prague, July 12-15, 1993, pp315-327.

Kemp, NH and Sears, WR, 1955, 'The Unsteady Forces due to Viscous Wakes in Turbomachinery', J. of Aeronautical Sciences, Vol. 22, No. 7, pp 478-483.

Kerrebrock, JL and Mikolajczak, AA, 1970, 'Intra – Stator Transport of Rotor Wakes and its Effect on Compressor Performance', ASME Jnl. of Engineering and Power, October 1990, pp 359-368.

Korakianitis, T, 1991, 'On the Propagation of Viscous Wakes and Potential Flow in Axial Turbine Cascades', ASME Paper No. 91-GT-373.

Kuroumaru, M, Inoue, M, Higki, T, Abd-Elkhalek, FA-E and Ikui, T, 1982, 'Measurement of Three Dimensional Flow Field Behind an Impeller by means of Periodic Multi-sampling with a Slanted Hotwire', Bulletin of JSME, Vol. 25, No. 209, pp 1674-1681, November 1982.

LaFleur, RS, Langston, LS and Sharma, OP, 1988, 'Flow Visualisation Investigation Through a Turbine Cascade with Incoming Streamwise Vortices', Unpublished Report, University of Connecticut, Storrs.

Lakshminarayana, B and Horlock, JH, (1963), 'Secondary Flow and Losses in Cascades and Axial-Flow Turbomachines', International Journal of Mechanical Sciences, Vol. 5, 1963.

Lewis, KL, 1993, 'The Aerodynamics of Shrouded Multistage Turbines', Ph.D. Thesis, Cambridge University, November 1993.

Lighthill, MJ, 1954, 'The Response of Laminar Skin Friction and Heat Transfer to Fluctuations in the Stream Velocity', Proc. of Royal Society, Vol. 224A, pp. 1-23.

Mayle, RE, 1991, 'The Role of Laminar – Turbulent Transition in Gas Turbine Engines', First IGTI Gas Turbine Scholar Lecture, ASME Paper No. 91-GT-261.

Meyer, RX, 1958, 'The Effect of Wakes on the Transient Pressure and Velocity Distributions in Turbomachines', ASME Jnl. of Basic Engineering, October 1958, pp 1544-1552.

Moore, J and Tilton, JS, 1988, 'Tip Leakage Flow in a Linear Turbine Cascade', ASME Jnl. of Turbomachinery, Vol. 110, pp 18.

Morphis, G and Bindon, JP, 1988, 'The Effects of Relative Motion, Blade Edge Radius, Gap Size on the Blade Tip Pressure Distribution in an Annular Turbine Cascade with Clearance', ASME Paper No. 88-GT-256.

Ni, RH and Sharma, OP, 1990, 'Using 3D Euler Flow Simulations to Assess Effects of Periodic Unsteady Flow Through Turbines', AIAA Paper No. 90-2357.

Ni, RH, 1982, 'A Multi Grid Scheme for Solving the Euler Equations', AIAA Journal, Vol. 20, No. 11, pp 1565-1571.

Parker, R and Watson, JF, 1972, ‘Interaction Effects between Blade Rows in Turbomachines’, Proc. of I. Mech. E., Vol. 186, No. 21

Pfeil, H, Herbst, R and Schroder, Th, 1982, ‘Investigation of Laminar-Turbulent Transition of Boundary Layers Disturbed by Wakes’, ASME Paper No. 82-GT-124.

Press, WH, Teukolsky, SA, Vetterling, WT and Flannery, BP, 1999, ‘Numerical Recipes in Fortran 77’, Cambridge University Press, Second edition.

Potts, I, 1987, ‘The Importance of S1 Stream Surface Twist in the Analysis of Inviscid Flow through Swept Linear Turbine Cascades’, I. Mech. E. Paper C258/87.

Rai, MM, 1987, ‘Navier-Stokes Simulations of Rotor-Stator Interactions using Patched and Overlaid Grids’, Jnl. of Propulsion and Power, Vol. 3, No.5, pp387-396.

Raj, R and Lakshminarayana, B, 1976, ‘Three-Dimensional Characteristics of Turbulent Wakes behind Rotors of Axial Flow Turbomachinery’, ASME Jnl. of Engineering for Power, April 1976, pp 218-228.

Saxer, AP and Giles, MB, 1993, ‘Predictions of 3D Steady and Unsteady Inviscid Transonic Rotor-Stator Interaction with Inlet Radial Temperature Non-Uniformity’, ASME Paper No. 93-GT-10.

Scherbakov, AP, Tischenko, AA and Dohler, SW, 1997, ‘The Loss Behaviour of Transonic Flow in Nozzle Cascades under the Influence of Unsteady Effects’, 2nd European Conference on Turbomachinery, Fluid Dynamics and Thermodynamics, Antwerpen, March 5-7, 1997.

Schlichting, H, 1966, ‘Boundary Layer Theory’, 6th edition, McGraw-Hill, New York.

Schlichting, H, 1987, ‘Boundary Layer Theory’, 8th edition, McGraw-Hill, New York.

Sharma, OP and Butler, TL, 1987, ‘Prediction of Endwall Losses and Secondary Flows in Axial Flow Turbine Cascades’, ASME Jnl. of Turbomachinery, Vol. 109, p229.

Sharma, OP, Butler, TL, Joslyn, HD and Dring, RP, 1985, ‘Three Dimensional Unsteady Flow in an Axial Flow Turbine’, AIAA Paper No. 83-1170, Jnl. of Propulsion, Jan-Feb. 1985, Vol. 1, pp29-38.

Sharma, OP, Ni, RH and Tanrikut, S, 1994, ‘Unsteady Flows in Turbines – Impact on Design Procedure’, AGARD Lecture Series ‘Turbomachinery Design using CFD’, May-June, 1994.

Sharma, OP, Pickett, GF and Ni, RH, 1990, ‘Assessment of Unsteady Flows in Turbines’, ASME Paper No. 90-GT-150, June 1990.

Sharma, OP, Renaud, E, Butler, TL, Milsaps, K, Dring, RP and Joslyn, HD, 1988, ‘Rotor-Stator Interaction in Multistage Axial Flow Turbines’, AIAA Paper No. 88-3013, AIAA/ASME/SAE/ASEE 24th Joint Propulsion Conference, Boston, July 1988.

Schulte, V, 1995, ‘Unsteady Separated Boundary layers in Axial Flow Turbomachinery’, Ph.D. Thesis, Cambridge University.

Sieverding, CH, 1984, ‘Recent Progress in the Understanding of Basic Aspects of Secondary Flows in Turbine Blade Passages’, ASME Paper No. 84-GT-78.

Sjolander, SJ and Amrud, KK, 1987, ‘Effects of Tip Clearance on Blade Loading in a Planar Cascade of Turbine Blades’, ASME Jnl. of Turbomachinery, Vol. 109, pp 237.

Smith, LH, 1955, ‘Secondary Flow in Axial-Flow Turbomachinery’, Transaction of ASME, Vol. 77, 1955.

Smith, LH, 1966, ‘Wake Dispersion in Turbomachines’, ASME Jnl. of Basic Engineering, 1966, pp 688-690.

Stauter, RC, Dring, RP and Carta, FO, 1991, ‘Temporally and Spatially Resolved Flow in a Two Stage Axial Flow Compressor: Part 1 – Experiment’, ASME Jnl. of Turbomachinery, Vol. 113, April 1991.

Takanashi, RK and Ni, RH, 1990, ‘Unsteady Euler Analysis of the Redistribution of an Inlet Temperature Distortion in a Turbine’, AIAA Paper No. 90-2262.

Valkov, T, 1995, ‘Control of the Unsteady Flow in a Stator Blade Row Interacting with Upstream Moving Wakes’, ASME Jnl. of Turbomachinery, Vol. 117, pp 97-105.

Van de Wall, A, Kadambi, JR and Adamczyk, JJ, 2000, ‘A Transport Model for the Deterministic Stresses Associated with Turbomachinery Blade Row Interactions’, ASME Paper No. 2000-GT-430.

Van de Wall, A, Kadambi, JR, Boyle, RJ and Adamczyk, JJ, 1995, ‘The Transport of Vortices through a Turbine Cascade’, ASME Paper No. 95-GT-240.

Wadia, AR and Booth, TC, 1982, ‘Rotor-Tip Leakage: Part II – Design Optimisation through Viscous Analysis and Experiment’, ASME Jnl. of Engineering for Power, Vol. 104, p 162.

Walker, GJ and Oliver, AR, 1972, ‘The Effect of Interaction between Wakes from Blade Rows in an Axial Flow Compressor on the Noise Generated by Blade Interaction’, ASME Jnl. of Engineering for Power, October 1972, pp 241-248.

Walker, PJ, 1987, ‘Blade lean in axial turbines: Model turbine Measurements and Simulation by Novel Numerical Method’, Ph.D. thesis, Cambridge university.

Wallis, AM, 1997, ‘The Control of Secondary and Leakage Flows in a Shrouded Multistage Turbine’, Ph.D. Thesis, Cambridge University.

Walraevens, RE, Gallus, HE, Jung, AR, Mayer, JF and Stetter, H, 1998, ‘Experimental and Computational Study of the Unsteady flow in a 1.5 Stage Axial Turbine with Emphasis on the Secondary Flow in the Second Stator’, ASME Paper No. 98-GT-254.

Walsh, JA and Gregory-smith, DG, 1987, ‘The Effect of Inlet Skew on the Secondary Flows and Losses in a Turbine Cascade’, Efficiency Prediction Improvement, I Mech. E, Paper No. C275/87.

Wang, HP, Olson, SJ, Goldstein, RJ and Eckert, ERG, 1995, ‘Flow Visualisation in a Linear Turbine Cascade of High Performance Turbine Blades’, ASME Paper No. 95-GT-7.

Yaras, MI and Sjolander, SA, 1990, ‘Prediction of Tip Leakage Losses in Axial Turbines’, ASME Paper No. 90-GT-154.

Yaras, MI and Sjolander, SA, 1992, ‘Effects of Simulated Rotation on Tip Leakage in a Planar Cascade of Turbine Blades Parts 1 and 2’, ASME Jnl. of Turbomachinery, Vol. 114, pp 652-667.

Appendix - A1: Uncertainty Analysis

Knowing the experimental uncertainty associated to a measurement allows the comparison to be made with the same measurement taken under different circumstances. The measurement of the same quantity by means of the same equipment but in different facilities will most probably not produce identical results. Differences are due to the variations in the calibration, environmental effects, and human factors. The results of a repeated experiment are nevertheless expected to remain within the interval around each measurement point. This interval is referred to as the experimental uncertainty. It is a measure of possible error made on a particular experiment.

The overall uncertainty of a measurement system can be calculated by examining the logarithmic derivative of the equation relating the unknown quantity to the measured variables. A maximum possible uncertainty can then be computed by simply summing the appropriate uncertainty terms (Arts *et. al.* (1994)). This approach also allows to verify the importance of each uncertainty term. The calculation of some important parameters measured in the experiment such as the uncertainty of velocity, density, mass flow rate, coefficient of stagnation pressure loss and isentropic efficiency are given below.

The flow at the inlet to the stage passes through a large inlet contraction fitted with honeycomb flow straighteners and is uniform. The axial velocity of the flow inlet to the turbine stage can then be calculated from the relation

$$V_{x1} = \sqrt{\frac{2(P_{01} - P)}{\rho}} \qquad \text{(A1.1)}$$

The uncertainty equation for the same is given as

$$\delta V_{x1} = \left\{ \left[\left(\frac{\partial V_{x1}}{\partial (P_{01} - P)} \right) \delta (P_{01} - P) \right]^2 + \left[\left(\frac{\partial V_{x1}}{\partial \rho} \right) \delta \rho \right]^2 \right\}^{\frac{1}{2}} \qquad \text{(A1.2)}$$

After replacing the partial derivatives, the equation A1.2 is given as

$$\delta V_{x1} = \left\{ \left[\left(\frac{\sqrt{2}}{\sqrt{\rho}\ 2\sqrt{(P_{01} - P)}} \right) \delta(P_{01} - P) \right]^2 + \left[\left(-\frac{\sqrt{2(P_{01} - P)}}{0.5\rho^{3/2}} \right) \delta\rho \right]^2 \right\}^{\frac{1}{2}} \qquad \text{(A1.3)}$$

By substituting typical values encountered in the experiment for ρ, (P_{01}-P) and the error in measuring these flow parameters with various measuring devices, one can calculate the uncertainty in measuring the axial velocity of the flow. Table A1.1 shows the estimated errors associated with each measurement with various measuring devices. The uncertainty in measuring the velocity is calculated as ± 0.128 m/sec.

Parameter	Measurement device	Approximate value	Variable error	$\delta x / x$ %
T	Load cell	270 Nm	0.5	0.18
Ω	Once per rev optical trigger	550 rpm	0.01	0.002
Δp	ZOC	1215pa	2pa	0.16
T_{01}	Thermocouple	300K	0.5K	0.17
Δp_0	Scanivalve, Pitots	1215 pa	3pa	0.25
Δp_0	Scanivalve, Pitots	1215 pa	3pa	0.25
P_{01}	Barometer	100000 pa	20pa	0.02

Table A1.1 Estimation of errors associated with the measurement of turbine isentropic efficiency

It can be observed from the equation A1.3 that the uncertainty in axial velocity measurement depends on the uncertainty of another parameter flow density. The density is calculated from the relation

$$\rho=(P/RT) \qquad \text{(A1.4)}$$

The uncertainty equation for the same can be given as

$$\delta\rho = \left\{\left[\left(\frac{\partial\rho}{\partial p}\right)\delta p\right]^2 + \left[\left(\frac{\partial\rho}{\partial T}\right)\delta T\right]^2\right\}^{\frac{1}{2}} \tag{A1.5}$$

After replacing partial derivatives, the equation A1.5 can be written as

$$\delta\rho = \left\{\left[\left(\frac{1}{RT}\right)\delta p\right]^2 + \left[\left(-\frac{P}{R\,T^2}\right)\delta T\right]^2\right\}^{\frac{1}{2}} \tag{A1.6}$$

The uncertainty in measuring the flow density is calculated as ± 0.002 Kg/m^3. Similarly, the mass flow rate passing through the test rig can be calculated from the relation

$$\dot{m} = \rho\, A\, V_{x1} \tag{A1.7}$$

and the corresponding uncertainty equation is given as

$$\delta\dot{m} = \left\{\left[\left(\frac{\partial\dot{m}}{\partial\rho}\right)\delta\rho\right]^2 + \left[\left(\frac{\partial\dot{m}}{\partial V_{x1}}\right)\delta V_{x1}\right]^2\right\}^{\frac{1}{2}} \tag{A1.8}$$

From the above equation, the uncertainty in measuring the mass flow rate passing through the turbine is calculated as ± 0.108 Kg/sec. Another important parameter, which characterises the flow losses in a blade row, is the coefficient of stagnation pressure loss. It is defined as

$$Y = \frac{P_{01} - P_{02}}{\frac{1}{2}\rho\left(\dfrac{V_{x1}}{\mathrm{Cos}74^0}\right)^2} \tag{A1.9}$$

In order to calculate the uncertainty of in estimating the loss coefficient, the equation A1.9 can be simplified to

$$Y = \frac{P_{01} - P_{02}}{6.581\rho\, V_{x1}^{\;2}} \tag{A1.10}$$

and the corresponding uncertainty equation is given as

$$\delta Y = \left\{\begin{array}{l}\left[\left(\dfrac{\partial Y}{\partial(P_{01} - P_{02})}\right)\delta(P_{01} - P_{02})\right]^2 + \left[\left(\dfrac{\partial Y}{\partial\rho}\right)\delta\rho\right]^2 + \\ \left[\left(\dfrac{\partial Y}{\partial V_{x1}}\right)\delta V_{x1}\right]^2\end{array}\right\}^{\frac{1}{2}} \tag{A1.11}$$

After replacing the partial derivatives, equation A1.11 becomes

$$\delta Y = \left\{ \begin{array}{l} \left[\left(\dfrac{1}{6.581\rho V_{x1}^{\;2}}\right)\delta(P_{01}-P_{02})\right]^2 + \left[\left(-\dfrac{(P_{01}-P_{02})}{6.581 V_{x1}^{\;2}\rho^2}\right)\delta\rho\right]^2 \\ + \left[\left(-\dfrac{(P_{01}-P_{02})}{6.581\rho V_{x1}^{\;3}}\right)\delta V_{x1}\right]^2 \end{array} \right\}^{\frac{1}{2}} \qquad \text{(A1.12)}$$

Typical values of ρ=1.2, V_{x1}=14 m/sec and (P_{01}-P_{02})=118 pa are used in the analysis. The uncertainties of individual δ(Vx1), $\delta(\rho)$ and $\delta(\dot{m})$ with δ(P_{01}-P_{02}) are estimated from the respective equations of density and velocity and from the accuracy of the measuring devices for (P_{01}-P_{02}) respectively. The uncertainty in measuring the coefficient of stagnation pressure loss in absolute frame of reference is calculated as $\pm$0.00145. The uncertainty in measuring the coefficient of stagnation pressure loss in relative frame of reference is calculated as $\pm$0.00205. The difference in uncertainty in two frames of reference is due to the use of different pressure measuring devices.

Another parameter, which characterises the performance of the turbine stage, is the isentropic efficiency, as defined by table 5.1. This is given as

$$\eta = \frac{\Omega T}{\dot{m} c_p T_{01}\left(1-\left(\dfrac{\overline{P_{0exit}}}{P_{01}}\right)^{\frac{\gamma-1}{\gamma}}\right)} \qquad \text{(A1.13)}$$

The uncertainty equation for the equation A1.13 is given as

$$\delta\eta = \left\{ \begin{array}{l} \left[\left(\dfrac{\partial\eta}{\partial\Omega}\right)\delta\Omega\right]^2 + \left[\left(\dfrac{\partial\eta}{\partial T}\right)\delta T\right]^2 + \left[\left(\dfrac{\partial\eta}{\partial\dot{m}}\right)\delta\dot{m}\right]^2 \\ + \left[\left(\dfrac{\partial\eta}{\partial T_{01}}\right)\delta T_{01}\right]^2 + \left[\left(\dfrac{\partial\eta}{\partial P_{0exit}}\right)\delta P_{0exit}\right]^2 + \left[\left(\dfrac{\partial\eta}{\partial P_{01}}\right)\delta P_{01}\right]^2 \end{array} \right\}^{\frac{1}{2}} \qquad \text{(A1.14)}$$

After substituting the respective values for the uncertainty of mass flow rate, density, angular speed, torque and temperature in equation A1.14 gives an overall uncertainty of $\pm$0.8% for the measurement of brake efficiency.

Appendix - A2: Integral Length Scale and Diffusion rate Measurements

It is useful to characterise the unsteady flow behind the stator for the ‘delta wing’ test configuration to analyse turbulence generating regions such as blade wake, hub and casing passage vortices. Two parameters, which characterise an unsteady flow, are the turbulence intensity and integral length scale. The turbulence intensity expresses the strength of the turbulence while the integral length scale assigns a spatial dimension to the turbulence structure, often identified as the average eddy size. The route to calculating the integral length scale lies through the correlation of two velocity signals or, if Taylor’s hypothesis is assumed as here, through the autocorrelation of a single velocity signal. The autocorrelation function (ACF) for a non-periodic signal is given by

$$\mathrm{ACF}(\tau) = \frac{\dfrac{1}{\Delta T}\displaystyle\int_0^{\Delta T} V(t)\,V(t+\tau)\,dt}{\dfrac{1}{\Delta T}\displaystyle\int_0^{\Delta T} V^2(t)\,dt} \tag{A2.1}$$

where ΔT is large compared to the period of the lowest frequency component of the signal with significant amplitude. To calculate the integral length scale, the autocorrelation function is integrated with respect to the time lag (τ) to obtain the integral time scale, which is multiplied by the mean velocity to give the length scale

$$\text{Integral length scale} = \overline{V}\int_0^{\infty} \mathrm{ACF}(\tau)\,d\tau \tag{A2.2}$$

Figure A2.1(e) shows an auto-correlation function calculated from the measured data. In practice, the integral length scale was evaluated from $\tau = 0$ to the value of τ at which ACF (τ) first becomes zero. One can see that the definition of integral length scale is only valid for random signals (those with no periodic order) because the autocorrelation of a periodic signal is itself periodic and does not tend to zero as τ increases. This condition imposes restriction on signals measured in turbomachinery environments, which have inherent periodic blade passing frequency content.

The method of Camp (1995) was used to overcome the restriction imposed by the definition of the integral length scale. In this method, the periodic components of the signal (those components at blade passing and its harmonics) were digitally removed by Fourier transforming the signal into a frequency domain and then setting the components at blade passing frequency and its four harmonics to zero amplitude. This method of calculating the turbulence intensity and integral length scale from the data set is shown schematically in figure A2.1. Figure A2.1(a) represents a schematic section of the raw velocity trace as recorded. The Fourier transform of this signal into the frequency domain reveals the spectrum, shown in figure A2.1(b). At this point the amplitudes of the components at blade passing frequencies and its harmonics are set to zero. This operation gives the ‘chopped’ spectrum shown in figure A2.1(c). This ‘chopped’ spectrum is now transferred back into the time domain to give the turbulent signal, shown in figure A2.1(d) from which turbulence intensity is calculated. The Fourier coefficients of the ‘chopped’ spectrum are also used directly using Wiener-Khinchin theorem (Press et. al. (1999)), to calculate the auto-correlation function (figure A2.1(e)) from which length scale is calculated.

Table A2.1 summarises the turbulence intensity and integral length scale results for the two test configurations (Rotating hub and Delta wing) at stator exit. The results are given at the centre of the free-stream, the stator blade wake and the stator hub and casing passage vortices regions. The length scales are non-dimensionalised by using the stator throat dimension. The length scales in the centres of the hub and casing passage vortices are 40% and 48% respectively for the rotating hub configuration. The corresponding length scale values for the delta wing configuration are 40% at both hub and 42% at the casing passage vortices. These results show that the eddy size of the casing passage vortex is larger than the hub passage vortex. This can be observed from the size of the high-turbulence intensity regions also. In the centre of the wake the magnitude of the length scales are given as 23% and 22% for the rotating hub and the delta wing configurations respectively. This indicates that the eddy size is smaller in the wake than in the passage vortices.

For a better understanding and complete documentation of turbulence flow, additional characterisation of the turbulent scales, turbulence kinetic energy (k) and local turbulence dissipation rates (ε) are needed. These quantities are particularly useful in computational modelling using k-ε or large eddy simulation (LES) models.

Figure A2.2 presents the spectral distribution of the tangential velocity in the centre of the stator passage vortex for the delta wing test configuration. This is a different type of plot to the one discussed in figure 6.17. The figure A2.2 presents the power spectral density based on wave number as a function of the wave number κ. Using Taylor's hypothesis, the wave number is defined as

$$\kappa = \frac{2\pi f}{\overline{V}} \tag{A2.3}$$

where f is the frequency in the power spectrum. The power spectral density based in the wave number domain can be expressed as

$$E(\kappa) = \frac{psd\ \overline{V} f}{2\pi} \tag{A2.4}$$

where psd is the power spectral density based on frequency. Figure A2.2 is useful in determining the dissipation rate and length scales within the turbulent flow. At low frequencies in the spectrum, the spectrum shows the energy associated with the large eddies in the flow. At high frequencies, the spectra shows the inertial sub-range (identified by the $\kappa^{-5/3}$ relationship) and for higher κ the spectra denote the dissipation range. At mid frequencies, the spectrum is related to the energy containing eddies in flow. At these mid frequency range, a spike in the spectra can be observed in figure A2.2 corresponding to the upstream delta wing vortex indicating the energy corresponding to the delta wing vortex.

Power spectral density distributions can also be used to determine energy dissipation rates when a -5/3 equilibrium inertial sub-range exists as explained by Hinze (1975), Burd and Simon (1999). This is done using the following equation, in which the dissipation rate is calculated by locating points (κ, E(κ)), on the spectral distributions that are tangent to a $\kappa^{-5/3}$ line:

$$E(\kappa) = \frac{18}{55} \, x \, 1.62 \, x \, \varepsilon^{2/3} \, \kappa^{-5/3} \tag{A2.5}$$

The selection of the coefficient 1.62 is consistent with Hinze (1975). In using this type of analysis, an assumption of isotropy is made. These turbulent flows are not expected to be isotropic in the larger scales, but in the inertial sub range, the turbulence is anticipated to be reasonably isotropic. Thus, this model is considered to give a suitable estimate of dissipation when a -5/3 relationship is visible in the power spectra.

Another important parameter characterising the turbulent flow is turbulent kinetic energy and it is calculated using the measured rms velocity fluctuations. To determine the turbulence kinetic energy, k, a formulation that assumes that the turbulence was isotropic was used. The turbulence kinetic energy is calculated as:

$$k = 1.5\,(V_x' + V_r' + V_\theta')^2 \tag{A2.6}$$

The knowledge of turbulence kinetic energy and the dissipation rate also provides a means of calculating the energy or dissipation length scale from Hancock and Bradshaw (1983). The length scale is given as

$$L_u = \frac{1.5(V')^3}{\varepsilon} = \frac{0.817\,(k)^{3/2}}{\varepsilon} \tag{A2.7}$$

A final scale that is deducted from the measurements is the Kolmogorov or micro scale of turbulence. It is given as

$$\eta = \left(\frac{\nu^3}{\varepsilon}\right)^{1/4} \tag{A2.8}$$

Table A2.2 summarises the results of turbulence kinetic energy, turbulence dissipation rate, dissipation length scale and Kolmogorov's microscales for the delta wing and rotating hub test configurations at the stator exit at various locations in the area traverse. The length scale and microscales of turbulence are non-dimensionalised using the stator chord.

It can be observed that the turbulence kinetic energy is very small in the freestream region and increases in the blade wake and the passage vortices regions as expected. In both the test configurations, it can also be seen that the hub passage vortices have large turbulence kinetic energies compared to the casing passage vortices. The turbulence dissipation rate varied from 78m^2/sec^3 in the freestream region to 4300m^2/sec^3 in the blade wake region and 7300m^2/sec^3 in the centre of the casing passage vortex regions for the rotating hub test configuration. The smallest eddies associated with the freestream have lower diffusion rates compared to the wake which has larger length scale. The passage vortex region has higher length scales than the wake as explained in table A2.1. It also corresponds to the higher diffusion rates. The trends are similar for the delta wing configuration. The higher dissipation rates in centre of wake and vortices regions are observed and very low dissipation rate in the freestream. The dissipation rates calculated here are for eddies in the inertial sub-range only and do not represent all the eddy sizes/frequencies possible. Nevertheless, it has demonstrated the wide range of diffusion rates (varying two orders of magnitude from 78 to 7300) that are possible at the measurement plane. The dissipation length scales also confirms the trend with lower length scales in the freestream and higher values in the viscous flow regions. In contrast the microscales of turbulence are high in the freestream region and lower in the viscous flow regions. These data of turbulence parameters is considered useful for CFD and turbulence modelling.

	Rotating Hub test configuration		*Delta wing test configuration*	
	'Tu' (%)	L_u/stator throat (%)	'Tu' (%)	L_u/stator throat (%)
Casing passage vortex	8.260	48.51	6.600	42.16
Hub passage vortex	8.860	40.71	8.580	40.67
Blade wake	6.140	23.03	7.420	22.13
Free-stream region	1.020	103.7	0.975	175.33

Table A2.1 Turbulence intensity and length scales at stator exit

	Rotating hub test configuration				*Delta wing test configuration*			
	k (m^2/sec^2)	ε (m^2/sec^3)	L_u/C (%)	η/C ($x10^4$)	k (m^2/sec^2)	ε (m^2/sec^3)	L_u/C (%)	η/C ($x10^4$)
Casing passage vortex	22.62	7328.2	8.53	1.819	18.38	7396.8	6.10	1.837
Hub passage vortex	26.46	5750.5	13.57	1.933	33.85	7009.2	16.10	1.862
Blade wake	17.85	4296.4	10.07	2.079	15.84	7813.8	4.63	1.812
Free-stream region	0.84	78.1	5.69	5.660	0.38	89.2	1.48	5.540

Table A2.2 Turbulence parameters and scales at stator exit

	Stator	**Rotor**
Flow Coefficient V_{x1}/U_m		0.3
Stage Loading		1.0
Stage Reaction		0.5
Mid-Span upstream Axial Gap(mm)		47.4
Hub-Tip Radius Ratio	0.8	0.8
Number of Blades	36	42
Mean Radius (m)	0.6858	0.6858
Rotational Speed (rpm)		550
Mid-span Chord (mm)	150	124
Mid-span Pitch-Chord Ratio	0.8	0.83
Aspect Ratio	1.01	1.22
Tip Clearance (mm)		1.4
Inlet Axial Velocity (m/s)	11.8	
Mid-span Inlet Angle (from axial)	0.0^0	4.8^0
Mid-span Exit Angle (from axial)	74.3^0	-74.4^0
Chord based Reynolds Number	$4.7x10^5$	$3.8x10^5$
Inlet Free-Stream Turbulence	0.25%	

Table 3.1 Turbine 1 - Turbine geometry and Test conditions

	Stator	**Rotor**
Flow Coefficient V_{x1}/U_m		0.35
Stage Loading		1.22
Stage Reaction		0.5
Mid-Span upstream Axial Gap(mm)		41.2
Hub-Tip Radius Ratio	0.8	0.8
Number of Blades	36	42
Mean Radius (m)	0.6858	0.6858
Rotational Speed (rpm)		550
Mid-span Chord (mm)	142.5	114.5
Mid-span Pitch-Chord Ratio	0.84	0.896
Aspect Ratio	1.07	1.33
Shroud Clearance (mm)		1.0
Inlet Axial Velocity (m/s)	13.85	
Mid-span Inlet Angle (from axial)	0.0^0	13.46^0
Mid-span Exit Angle (from axial)	71.03^0	-70.86^0
Chord based Reynolds Number	$5.24x10^5$	$4.12x10^5$
Inlet Free-Stream Turbulence	0.25%	

Table 3.2 Turbine 2 : 3D Turbine geometry and Test conditions

Traverse location	*Measurement type*	*Probe type*	*Number of pitches*	*Percent span measured*	*Number of pitchwise points*	*Number of spanwise points*	*Test configuration*
0	Steady flow field	5-hole	1 stator	1.11 – 34.0	21	26	3, 4
0	Unsteady flow field	SSHW	1 stator	5.84 – 97.7	1	16	4
1	Steady flow field	5-hole	2 stator	3.05 – 95.6	81	26	1,2,3,4
1	Unsteady flow field	3 hot-wire	1 stator	6.82 – 96.0	21	18	1,3,4
1	Unsteady flow field	SSHW	1 stator	2.96 – 95.0	37	24	2,4
3	Steady flow field	5-hole	1 rotor	6.56 – 96.5	45	23	1,2,3,4
3	Unsteady flow field	3 hot-wire	1 rotor	3.94 – 94.5	45	25	1,3,4
4	Steady flow field	3-hole	2 stator	0.50 – 99.2	43	28	2
4	Unsteady flow field	SSHW	1 stator	2.10 – 97.1	21	28	2
4	Unsteady flow field	Kulite	1 stator	4.20 – 98.7	1	42	2

Table 3.3 Summary of experimental traverse measurements

Positioning	*Accuracy – Stationary frame*	*Accuracy – Rotating frame*
Circumferential	± 0.017 degrees	± 0.025 degrees
Radial	± 0.25 mm	± 0.25 mm
Yaw	± 0.10 degrees	± 0.10 degrees

Table 3.4 Probe positioning accuracy

Calculation type	*Boundary layers*	*Mixing length factor*	*Smoothing factor*	*FACSEC*	*Number of time steps*
Steady	Fully turbulent	0.03 (blade) 0.02 (walls)	0.001	0.8	15000
Unsteady	Fully turbulent	0.03(blade) 0.02(walls)	0.001	0.8	> 200, 000

Table 3.5 Solver parameters for steady and unsteady calculations

Calculation Type	*Solver*	*Number of grid points (I * J * K)*	*Total number of nodes*	*Grid expansion ratio and maximum value*	*Other comments*
Test case – 1 Stator + Rotor	MULTIP	34 * 91 * 45 34 * 97 * 45	287,640	1.3, 20 – Spanwise 1.2, 15 – Pitchwise	Steady calculation
Test case – 1 Stator + Rotor	UNSTREST	34 * 91 * 45 34 * 97 * 45	287,640	1.3, 20 – Spanwise 1.2, 15 – Pitchwise	Unsteady calculation
Test case – 2 Stator + Rotor	MULTIP	41 * 92 * 45 – Stator 41 * 99 * 45 - Rotor	352,395	1.3, 20 – Spanwise 1.2, 20 – Pitchwise	Steady calculation
Test case – 2 Stator + Rotor	UNSTREST	61 * 92 * 45 – Stator 61 * 99 * 45 - Rotor	710,711	1.3, 10 – Spanwise 1.2, 5 – Pitchwise	Unsteady calculation
Test case – 4 DW + Stator + Rotor	MULTIP	61 * 111 * 45 – DW 61 * 92 * 45 – Stator 61 * 105 * 45 – Rotor	845,460	1.3, 5 – Spanwise 1.2, 5 – Pitchwise	Steady calculation
Test case – 4 DW + Stator + Rotor	UNSTREST	82 * 111 * 45 – DW 82 * 92 * 45 – Stator 82 * 105 * 45 – Rotor	1,136,520	1.3, 5 – Spanwise 1.2, 5 – Pitchwise	Unsteady calculation
Test case – 4 DW + Stator + Rotor	UNSTREST	61 * 92 * 45 – DW 61 * 92 * 45 – Stator 61 * 105 * 45 – Rotor	1,075,369	1.3, 5 – Spanwise 1.2, 5 – Pitchwise	Unsteady calculation – no viscous terms in stator and rotor
Delta wing	MULTIP	41 * 91 * 32 - DW	120,704	1.3, 20 – Spanwise	Steady calculation

Table 3.6 Summary of the steady and unsteady numerical simulations

Flow Coefficient	$\phi = \dfrac{V_{\infty_{in}}}{U_m}$
Mass flow rate	$\dot{m} = \rho_{in} V_{\infty in}\, \pi\left[\left(r_{tip} - \delta^*_{tip}\right)^2 - \left(r_{hub} + \delta^*_{hub}\right)^2\right]$
Rotor mean exit stagnation pressure	$\overline{P_{0exit}} = \dfrac{\int_{hub}^{tip} \rho_{exit} V_{x\,exit} P_{0exit}\, 2\pi r\, dr}{\int_{hub}^{tip} \rho_{exit} V_{x\,exit}\, 2\pi r\, dr}$
Stage pressure coefficient	$\dfrac{P_{01} - \overline{P_{0exit}}}{\rho_{in} U_m^2}$
Stage loading coefficient	$\dfrac{\Omega T/\dot{m}}{U_m^2}$
Total-Total Efficiency	$\eta = \dfrac{\Omega T}{\dot{m} c_p T_{01}\left(1 - \left(\dfrac{\overline{P_{0exit}}}{P_{01}}\right)^{\frac{\gamma-1}{\gamma}}\right)}$

Table 5.1 Definitions of stage performance variables

		Datum	*Rotating Hub*	*Delta Wing*
Delta wing exit	Yaw angle (deg.)	-	-	-0.0677
	$Y_{measured}$	-	-	0.0077
	Y_{total}	-	-	0.0077
	% of Datum stator loss	-	-	36.7
		-	-	
Stator exit	Yaw angle (deg.)	74.3311	73.0695	72.7898
	$Y_{measured}$	0.0210	0.0246	0.0375
	% of Datum	100.0	117.1	178.6
Additional loss	Total	-	17.1	78.6
(% of Datum)	Boundary layer skew	-	17.1	17.1
	Delta wing	-	-	36.7
	Loss due to vortex transport	-	-	24.8

Table 6.1 Area integrated values at stator exit

		Rotating Hub	*Delta Wing*
Rotor exit	Yaw angle (deg.)	-70.6083	-69.96
	$Y_{measured}$	0.0691	0.0687
	% of Datum	100.0	92.0

Table 6.2 Area integrated values at rotor exit

	Radially stacked turbine	3-D turbine
Hub boundary layer thickness $(\delta^*/h)_{hub}$	0.0060	0.0080
Casing boundary layer thickness $(\delta^*/h)_{casing}$	0.0069	0.0077
Flow Coefficient (ϕ)	0.30	0.35
Reaction	0.50	0.50
Stage Loading ($\Delta h_0/U_m^2$)	1.050	1.196
Total-total efficiency (%)	88.30	91.08
Stator stagnation pressure loss coefficient	0.0378	0.0210
Rotor stagnation pressure loss coefficient	0.0774	0.0498

Table 8.1 Comparison between the three dimensional turbine stage and a radially stacked turbine stage