FINITE VOLUME BASED NUMERICAL INVESTIGATION ON THE EFFECTS OF INTERNAL COOLING AND FILM COOLING ON THE THERMAL PERFORMANCE OF GAS TURBINE BLADES

SUFIAN FARID SHAKER

UNIVERSITI SAINS MALAYSIA

2010

FINITE VOLUME BASED NUMERICAL INVESTIGATION ON THE EFFECTS OF INTERNAL COOLING AND FILM COOLING ON THE THERMAL PERFORMANCE OF GAS TURBINE BLADES

By

SUFIAN FARID SHAKER

Thesis submitted in fulfillment of the
requirements for the degree
of the Master of Science

May 2010

ACKNOWLEDGEMENTS

Firstly, I dedicate this work to the first teacher Prophet Muhammad, Peace be upon him, then to my parents who have always encouraged, motivated and inculcated in me the moral values to lead a peaceful life. I would like to express my sincere thanks to my supervisor, Assoc. Prof. Dr. Mohd Zulkifly Abdullah for his precious guidance, advice, support, assistance and encouragement throughout my candidature period. Also, special thanks to Dr. Kamarul Arifin Ahmad for his help and advice to handle the research work. Special thanks also must be given to all academic and technical staff especially Mr. Abdul Hisham**,** Mr. Mohd Shahar and Mr. Mohamad Zihad for their help on my research work. Special thanks must be given to Mr. M. Abdul Mujeebu for his help and encouragement during all the obstacles and challenges to this thesis. I would like to extend my gratitude to my colleagues, Mr. Azmi, Mr. Khalil, Mr. Muhad Rozi and my friends as well for their moral support and help during my research work. I would also like to express my gratitude to my brothers, Mr. Nathir and Mr. Muhammad and my sisters and also other family members for their continuous support in my life. Their support enabled me to concentrate fully on my studies without any stresses. I am grateful to the University Science Malaysia for the opportunity to complete my research work and I am proud to be one of the postgraduate students of this university. Last but not the least, my sincere thanks, compliments and regards to everyone who helped and supported me, in one way or another.

SUFIAN FARID SHAKER

May 2010

TABLE OF CONTENTS

CHAPTER 3 – METHODOLOGY

CHAPTER 4 – RESULTS AND DISCUSSION

LIST OF TABLES

LIST OF FIGURES **Page**

LIST OF SYMBOLS

C_p	specific heat capacity (J/kg.K)
d	the film cylindrical holes diameter (mm)
H	the blade span height (mm)
k_s	thermal conductivity (W/mK)
p	the distance between the film cooling holes
P_{in}	the inlet total pressures of the turbine stage (atm)
P_{ex}	exit static pressures of the turbine stage (atm)
$P_{R,ci}$	coolant injection pressure ratios
$P_{c,in}$	the total coolant inlet pressure (atm)
$P_{c,ex}$	exit static coolant pressure (atm)
T	passing time period (s)
t	time (s)

Greek symbols

ρ	density (kg/m^3)
μ	viscosity (kg/ms)
τ	shear stress
ω	angular velocity (rpm)

C	velocity magnitude
U	velocity magnitude in x-direction
V	velocity magnitude in y-direction
W	velocity magnitude in z-direction
e	Internal energy

KAJIAN BERANGKA BERDASAKAN ISIPADU TERHINGGA KE ATAS KESAN PENYEJUKAN DALAMAN DAN PENYEJUKAN FILEM KE ATAS PRESTASI TERMA BAGI BILAH TURBIN GAS

ABSTRAK

Di dalam enjin gas turbin, tren terkini adalah untuk memaksimumkan suhu masukan untuk mendapat keberkesanan terma tertinggi sekali gus meningkatkan keluaran. Akibatnya, permukaan bilah termasuk pinggir depan, sisi-sisi penghisap dan tekanan terdedah kepada tekanan haba lampau dan ia memerlukan penyejukan yang mencukupi untuk mengekalkan keselamatannya. Tesis ini mempersembahkan penyiasatan berangka dua dan tiga dimensi (2D dan 3D) terhadap kesan-kesan penyejukan dalaman dan tipisan ke atas perilaku terma bilah gas turbin dengan menggunakan kod komersial pengkomputeran dinamik bendalir, FLUENT™. Geometri bilah 3D yang mencontohi bilah sebenar McDonnell Douglas (A-4 Skyhawk) dijanakan di dalam pra-pemprosesan GAMBIT. Empat konfigurasi penyejukan iaitu 1) Salur dalaman U-liku tunggal, 2) Salur dalaman U-liku berganda, 3) Empat barisan penyejukan tipis dengan salur dalaman U-liku dan 4)Lapan barisan penyejukan tipis dengan salur dalaman U-liku, telah siap disimulasikan. Jaringan segi tiga baji tak tersusun telah digunakan untuk kesemua model bilah. Bahan yang dipertimbangkan di dalam analisis ini adalah *stainless steel alloy-AL 6XN* yang mana mempunyai rintangan tinggi terhadap haba. Model gelora diwakili oleh model k- ω pengangkutan ricih-tegasan (SST) dan aliran dianggap mempunyai 10% keamatan aliran-bebas gelora. Pekali pemindahan haba, jumlah taburan suhu, tekanan statik dan halaju vektor disiasat. Kesan nisbah

penyuntikan penyejuk tekanan ($P_{R.ci}$) ke atas taburan suhu juga disiasat. Keputusan menunjukkan pekali haba tentu dengan penyejuk tipisan adalah lebih baik daripada tanpa penyejuk tipisan. Berdasarkan kepada profil suhu ramalan, ia dapat diperhatikan bilah yang mempunyai lapan baris penyejuk tipisan dengan salur dalaman U-liku adalah yang terbaik untuk prestasi penyejukan. Selanjutnya, penambahan dalam $P_{R.ci}$ membantu mengurangkan suhu dan membekalkan lapisan penyejukan terbaik. Persembahan keputusan simulasi ini ditentu sahkan dengan keputusan penyelidik-penyelidik terdahulu. Model 2D yang mengkaji aliran tak mantap ke atas bilah rotor dengan menggunakan model berkala pada tiga kelajuan putaran iaitu 1800, 2550 dan 3000 ppm juga ditunjukkan. Keputusan menunjukkan suhu bilah rotor merosot sedikit apabila kelajuan putaran meningkat kerana kesan pecutan aliran. Kesimpulannya bahawa model yang disarankan ini dan juga simulasi FLUENT™ adalah cukup kuat untuk menangani permasalahan penyejukan di dalam bilah gas turbin.

FINITE VOLUME BASED NUMERICAL INVESTIGATION ON THE EFFECTS OF INTERNAL COOLING AND FILM COOLING ON THE THERMAL PERFORMANCE OF GAS TURBINE BLADES

ABSTRACT

The inlet temperature of modern gas turbine engines has been increased to achieve higher thermal efficiency and increased output. The blade surfaces including the leading edge, suction side and pressure side are exposed to the most extreme heat loads, and therefore, must be adequately cooled to maintain safety. This thesis presents steady computational predictions for a three-dimensional numerical investigation for different cooling arrangement of turbine blade model with both the internal cooling and combines cooling (film cooling) techniques. The computational predictions were achieved using FLUENT^{TM}. The dimension and the airfoil shape are generated by GAMBIT similar to the actual McDonnell Douglas (A-4 Skyhawk) blade with the same scale. The unstructured Triangular Pave mesh is used for all the blade models. The material considered in the analysis is stainless steel alloy-AL 6XN which is highly resistant to temperature. Turbulence was represented using the k- ω shear-stress transport (SST) model, and the flow was assumed to have a free-stream turbulence intensity of 10%. The heat transfer coefficient, total temperature distribution, static pressure and velocity vector was investigated. The CFD heat transfer coefficient predictions compared well with the previous work. The results showed that heat transfer coefficient was higher with film cooling compared to without film cooling. For all the cases, the temperature distributions over the blade surface are obtained and presented. From the results, the CFD simulation has

predicted that the blade with three rows of film cooling holes in the suction side, three rows in the pressure side and two rows in the leading edge with enternal channel is the best in terms of cooling performance. Besides these cases, the effect of coolant injection pressure ratio on temperature distribution $P_{R,ci}$ was investigated. Results showed that the increase in $P_{R,ci}$ has led to reduction in the temperature and moreover the lateral spreading facilitated the best coolant layer. This CFD code is also capable of handling unsteady flows. The periodic model was employed to predict the unsteady flow behavior during operation by a two-dimensional numerical investigation for details of film cooling blade at three rotational speeds of 1800, 2550 and 3000 rpm. Results showed that the temperature of the rotor blade decreased slightly with the increase in the rotation speed due to the effect of flow acceleration.

CHAPTER ONE

INTRODUCTION

1.1 Theory of Gas Turbine Engines

The turbine engine operates on the Brayton cycle, a constant pressure cycle containing the same four basic operations as the Otto cycle, but accomplishing them simultaneously and continuously so that an uninterrupted flow of power from the engine results. Ambient air is drawn into the inlet section by the rotating compressor. The compressor forces this incoming air rearward and delivers it to the combustion chamber at a higher pressure than the air had at the inlet. The compressed air is then mixed with fuel that is sprayed into the combustion chamber by the fuel nozzles. The fuel and air mixture is then ignited by electrical igniter plugs similar to spark plugs. This ignition system is only in operation during the starting sequence, and once started, combustion is continuous and self-sustaining as long as the engine is supplied with the proper air-fuel ratio. Only about 25 percent of the air is used for combustion. The remaining air is used for internal cooling and pressurizing.

The three classifications of gas turbine engines are turbojet, turbo - shaft, and ramjet. The term "turbo" means "turbine." Therefore, a turbo shaft engine is one which delivers power through a shaft. The gas turbine engines used to power aircraft are turbo-shaft power plants. The energy produced drives the power-shaft. Energy is generated by burning the fuel-air mixture in the engine and accelerating the gas tremendously. These high-velocity gases are directed through turbine wheels which convert the axial movement of the gas to a rotary motion. This rotary power is used to drive a power shaft, which drives a propeller or a rotor transmission. The turbine engines in the Army inventory are of the free-power turbine design, as shown in Figure 1.1. In this engine, nearly two-thirds of the energy produced by combustion is extracted by the gas producer turbine to drive the compressor rotor. The power turbine extracts the remaining energy and converts it to shaft power, which is used to drive the output shaft of the engine. The gas then exits the engine through the exhaust section to the atmosphere. The advantages of gas turbines compared to reciprocating engines are , higher power-to-weight ratio, less maintenance, less drag, easy cold weather starting and low oil consumption.

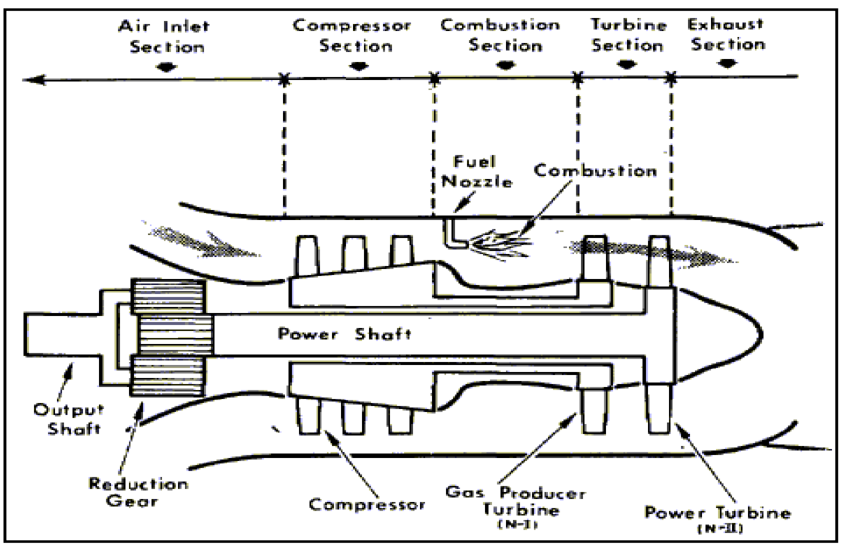

Figure 1.1: Schematic of a turbo- shaft engine (http://www.globalsecurity.org).

Gas turbines are used in many different applications ranging from power plants to ship engines aircrafts and even heart assist devices. This wide range of usage makes it very desirable to achieve maximum efficiency. More efficient engines will require less fuel leading to less expensive plane tickets and electricity. A small increase in efficiency amounts to a large savings in energy universally. The question arises of how to effectively increase the efficiency of these engines. One area that can be improved in a gas turbine engine is the turbine section. If we see the working principle, as long as the turbine produces more energy than the compressor, the engine works. Due to the thermodynamics of a gas turbine engine, the higher the combustion temperature is, the more efficient the engine. Thus it is desireable to increase the combustion temperature as much as possible, but there is a limiting factor. The materials that make up the turbine section cannot be allowed to heat to the point of failure or melting. Selection of temperature resistant materials and effective cooling of the turbine blades are the feasible options to tackle this issue.

Over the last decade the temperature has risen from 1500 K to 1750 K in some high-performance units. With this increase of the temperature only about 25% can be attributed to improved alloys. New materials, such as ceramics, could help increase this maximum temperature even more in the future. However, most of the recent improvements in inlet temperature come from better cooling of the blades and a greater understanding of the heat transfer and the three dimensional temperature distribution in the turbine passage. Higher gas temperature generally causes increased blade temperature and greater temperature gradients, both of which can have a detrimental effect on service life. As of today improvements in computational techniques in turbomachines have been attempted by industrial researchers because the numerical approaches are quite advantageous in comparison with experimentation, due to its ease of modeling, relatively complicated geometry and unsteady flow nature. The present investigation deals with the numerical modeling of channeled cooling of gas turbine blades, with and without considering the film cooling.

1.2 Cooling of Gas Turbine Blades

1.2.1 History of Blade Cooling

In 1929, Brown Boveri Corporation is reported to have manufactured turbine blades intended for air cooling (Bathie, 1996). The first turbo-jet engine that had cooled blades and that actually was in service was the German Junker Jumo 004, which flew shortly before the end of World War II. The cooled blades were simply made from a bent metal sheet, and thus, were hollow inside (Wilson, 1984). The purpose of applying cooling to the Junker Jumo was not to raise the turbine inlet temperature, but instead to allow the use of non-heat-resistant materials. After World War II, however, research and development in the domain of gas turbine cooling and materials was performed in order to raise cycle efficiency. According to Rohsenow (1955), the National Advisory Committee for Aeronautics (NACA) published several unclassified reports on experimental work with blade cooling in 1947. During the 1950s and 1960s, extensive effort was made in the domain of gas turbine cooling and depending on the source, it appears that from the 1960s or the 1970s onward, cooling has been an integrated part of commercially available gas turbines (Bathie,1996) and (Geipel et al. 1998). As a rule of thumb, the turbine-inlet temperature of modern gas turbines has, on average, been increasing at a rate of 20^0C per year since the 1960s. Approximately half of the increase is due to improved cooling methods, and the other half is due to improved high-temperature materials. An illustration of previous and expected future development is given in Figure 1.2. Compressor discharge air is still the coolant most commonly used. In the early 1960s, however, General Electric initiated development on a water cooled gas turbine, and the first laboratory model was tested in 1973 (Bathie, 1996). The system was reported to be still under development in 1982 by Wilson (1984) but appears to have come to an end without being launched on the market. Further developments in this area will be described in chapter 2.

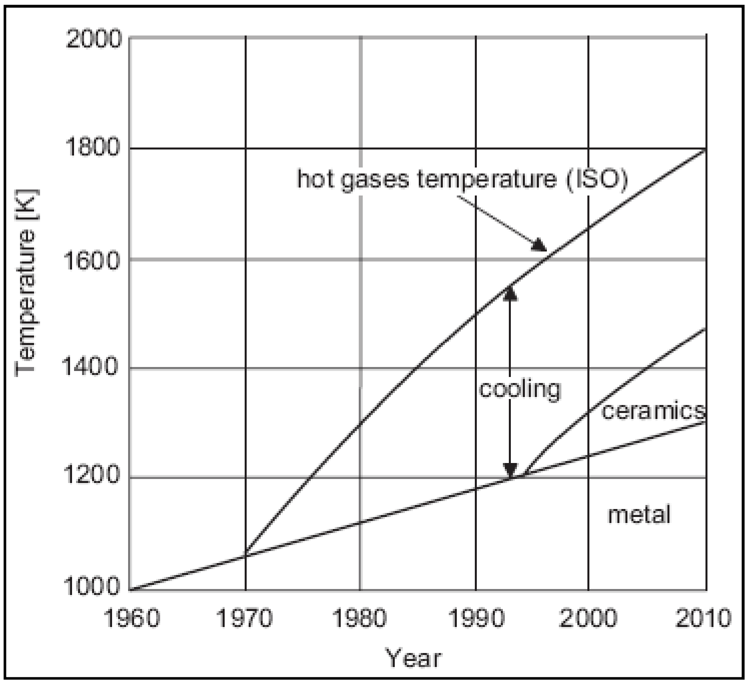

Figure 1.2: The development over time of gas-turbine materials and hot gases temperature levels and the impact of cooling on this development (Jordal, 2001).

1.2.2 Cooling Techniques

Cooling technologies can be categorized as open-loop and closed-loop cooling. In open-loop cooling when the coolant, after having absorbed heat, is injected into and mixed with the main flow, which means that the heat absorbed by the coolant is returned to the hot gases. In closed-loop, the coolant passes through the blade or vane, absorbs heat and rejects it outside of the expansion process. Open-loop air cooling is the most common way to cool a gas turbine. Basically four ways of cooling blades and vanes have been reported in the literature Jordal (2001): convection, impingement, film and transpiration cooling as illustrated in Figure 1.3.

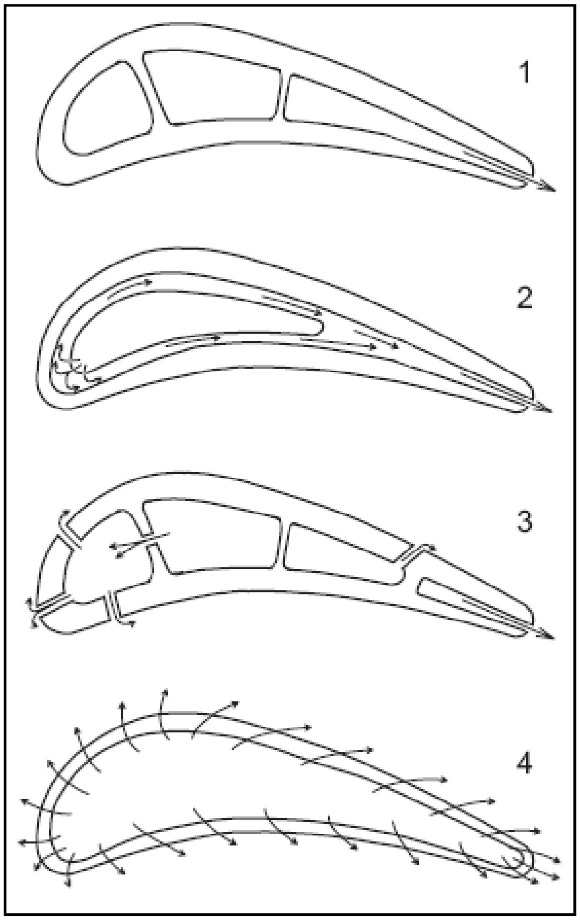

Figure 1.3: Principles of the four cooling methods: 1. convection cooling,
2. impingement cooling, 3. film cooling, 4. transpiration cooling (Jordal, 2001).

In air cooled gas turbines, a combination of film, convection and impingement cooling is usually applied. For closed-loop cooling, only convection and impingement cooling can be applied.

Convection cooling was the earliest method to be applied. The coolant flows in channels within the blade and is heated as it cools the blade. High velocity is required when air is employed as the coolant, and to increase the heat transfer, turbulence can be enhanced with different kinds of fins and ribs in the channels, but at the cost of more difficult manufacturing. The coolant is usually discharged at the trailing edge and mixed with the main flow, but can also be discharged outside of the blade or vane. Convection cooling technology has developed so that today, the coolant can be directed inside the blade to where it is most needed, i.e. at the leading and trailing edges.

Impingement cooling is a very efficient way to cool locally, usually on the blade leading edge. The air is directed radially through a central core in the blade, and then turned to the axial direction and made to impinge through small holes onto the inside of the blade. Impingement cooling is usually employed on stator vanes, but the method can be adapted to rotor blades as well.

Transpiration cooling is sometimes claimed to be the most efficient cooling method. The coolant is forced through the porous blade surface and creates a continuous film that protects the surface, at the cost of a rough surface that might decrease turbine efficiency. Manufacturing and materials problems have led to the fact that transpiration cooling still remains a hypothetical cooling method. However, it is sometimes argued that the development of the "shower-head" film cooling design at the leading edge of certain blades and vanes, is very close to local transpiration cooling.

Film cooling is an efficient way to protect the blade surfaces from the surrounding hot gases. The cooling air passes through holes in the blade surface and forms a protective film of relatively low temperature. Film cooling in its various forms has become one of the most important and relied upon cooling techniques for the hot sections of propulsion and energy gas turbines. It represents one of the few game changing technologies that have allowed the achievement of today’s high firing temperature, high efficiency gas turbine engines. Because of its high importance and widespread application, research on its various aspects has seen a tremendous increase in the last ten to fifteen years.

Film cooling is accomplished by injecting, from within the blade, a thin film of cold air along the blade surface separating the blade from the hot gas stream flow. Most of this air issues out of tiny (film-cooling) holes into the high temperature boundary layer on the blade surface, in an effort to form a cooler layer between the hot gas stream and the blade surface. Figure 1.4 is a simplified illustration of the complex heat transfer phenomena in the gas turbine path.

Though the film cooling technology has enabled the development of today's high-temperature gas turbines, it has some drawbacks. The film increases the boundary layer thickness, which results in higher blade profile losses and, worse, if the coolant is injected with a very high velocity, it penetrates the boundary layer and decreases the protection of the blade. The source of the cold coolant air is bleed-off air from the last stage of the compressor section. This high pressure air bypasses the combustor section of the engine, and in so doing, is maintained at much lower temperatures than the core turbine flow. This bleed-off air, however, is removed from the core mass flow and subtracts from the overall thrust of engine. Obviously, the engine designer wants to minimize the amount of bleed-off air required to cool the blade. From this, it is apparent that a good understanding of not only how film cooling schemes affect heat transfer, but also an idea of what parameters control the level of effectiveness provided, is needed. The ultimate purpose of the turbine designer is to maintain or increase the high level of turbine performance, and reduce, if possible, the amount of coolant flow needed to achieve this purpose.

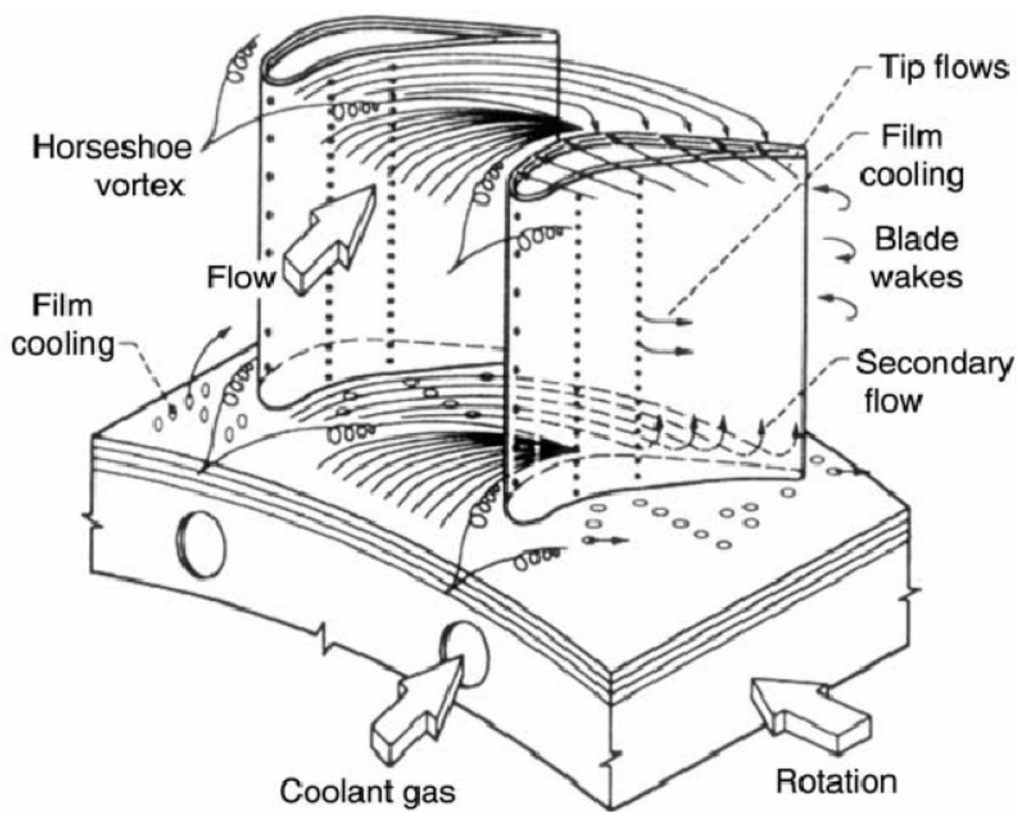

Figure 1.4: Complex heat transfer phenomena in the gas turbine path (Garg, 2002).

Film cooling is typically applied on the hottest parts of the gas turbine, e.g. the first-stage blades and vanes. Holmer (2000) reported that the row of film cooling holes might consist of 20-30 holes of a diameter of about 1 mm. The number of rows on a blade is sometimes just a few, and sometimes more than 10.

1.3 Problem Statement

The gas turbine blades have been facing the challenge of damaging. The resulting damage is often seen as localized burning or melting of the airfoil. Moreover, turbine blade failures account for 25.5% of gas turbine failures. Turbine blade oxidation, melting, corrosion, erosion and the forces arising from high rates of rotation are normally a long time process with material losses occurring slowly over a period of time. However, damage resulting from impact by a hot gas (fluid) and high pressure come from the combustion chamber.

In a high pressure turbine, first stage blade is one component that is extremely exposed to the hot gas. The gas turbine blade cooling is necessary to reduce maintenance cost and extend the life of blade which is one of the important parts of gas turbine. A gas turbine blade will be capable of effectively reducing creep damage by forming a cooling through hole to cool a target area. Significant fatigue damage of the blade surface is predicted based on analysis of temperature distribution on the gas turbine blade. Also the thermal efficiency and power output of gas turbines increase with increase of gas turbine inlet temperature. For all these reasons, the need for enhanced research to develop improved cooling techniques for gas turbine blades is quite obvious.

1.4 Scope of the Present Work

This research focuses on analysis of cooling performance of gas turbine blades by two different cooling methods namely, internal cooling and combined cooling (internal cooling and external cooling). Internal cooling involves cooling channels inside the blade. In external cooling which is also known as film cooling, the cooling fluid is discharged through tiny holes distributed on the blade surface. Both these techniques are simulated by a three dimensional model for analyzing the steady state heat transfer coefficient and fluid flow over the gas turbine blades. Internal cooling has been modeled and simulated by two cases; one U-bend tube channel and two U-bend tube channels. The external cooling has been modeled and simulated by two cases; four rows film cooling with internal channel and eight rowes film cooling with internal channel. It is interesting to note that all of the cases are modeled by the same airfoil geometry. Moreover, by means of an unsteady state two dimensional model, the effect of rotation speed is studied with three different rotation speeds 1800, 2550 and 3000 revolution per minute (rpm). The shear-stress transport (SST) k- ω model is used as the turbulence model. The Computational Fluid Dynamics (CFD) software, FLUENT® 6.3.2 is used for the simulation.

1.5 Objectives

The main aims and objectives of this work are:

- To study through simulation, the effect of internal cooling on the temperature distribution of the blades during operation.
- To study the effect of combined internal and external cooling on blade temperature distribution
- To investigate the influence of number and position of the channels and film cooling system on blade temperature distribution.
- To investigate the heat transfer coefficient with and without film cooling.
- To investigate the effect of rotation on the blade with varying rotation speeds.
- To obtain the static pressure, velocity vector and total temperature distribution during operation.

CHAPTER TWO

LITERATURE REVIEW

2.1 Introduction

Substantial researches have been carried out in the area of gas turbine blade cooling and a comprehensive review of all those works is beyond the scope of this thesis. However, as part of the present study a thorough review of the previous works on film cooling, especially those deal with the problems of heat transfer and fluid flow, both numerical and experimental, have been made. The literature review is presented as three parts; 1) Effect of flow and film cooling on the blade. 2) Effect of heat transfer and film cooling on the blade. 3) Thermal performance (cooling efficiency) on the blade.

2.2 Effect of Flow and Film Cooling on the Blade

One of the widely used blade cooling techniques is film cooling. Film cooling is accomplished by injecting, from within the blade, a thin film of cold air along the blade surface separating the blade from the hot free stream gases. This film provides an insulating layer and has the effect of protecting the blade from high temperatures of the hot free stream flow. In the area of film cooling, substantial works, both experimental and numerical have been carried out by many researchers. Film-cooling effectiveness from shaped holes on the near tip pressure side and cylindrical holes on the squealer cavity floor was investigated by Mhetras et al. (2008). The pressure side squealer rim wall was cut near the trailing edge to allow the accumulated coolant in the cavity to escape and cool the tip trailing edge. Effects of varying blowing ratios (P_{in}/P_{ex}) and squealer cavity depth were also examined on film-cooling effectiveness. The film-cooling effectiveness distributions were measured on the blade tip, near tip pressure side and the inner pressure side and suction side rim walls using pressure sensitive paint technique. The internal coolant supply passages of the squealer tip blade were modeled similar to those in the GE-E3 rotor blade with two separate serpentine loops supplying coolant to the film-cooling holes. Two rows of cylindrical film-cooling holes were arranged offset to the suction side profile and along the camber line on the tip. Another row of shaped film-cooling holes was arranged along the pressure side just below the tip. The average blowing ratio of the cooling gas was controlled to be 0.5, 1.0, 1.5, and 2.0. A five-bladed linear cascade in a blow down facility with a tip gap clearance of 1.5% was used to perform the experiments. The free-stream Reynolds number, based on the axial chord length and the exit velocity, was 1,480,000 and the inlet and exit Mach numbers were 0.23 and 0.65, respectively. A blowing ratio of 1.0 was found to give best results on the pressure side, whereas the tip surfaces forming the squealer cavity gave best results for blowing ratio of 2.0. Results showed high film cooling effectiveness magnitudes near the trailing edge of the blade tip due to coolant accumulation from upstream holes in the tip cavity. A squealer depth with a recess of 2.1 mm caused the average effectiveness magnitudes to decrease slightly as compared to a squealer depth of 4.2 mm.

Choi et al. (2008) investigated the film cooling and heat transfer on the cutback trailing edge of a turbine blade with slot ejection. The internal test geometry was similar to the crossover impingement hole design used in modern gas turbine blades for trailing edge cooling. A liquid crystal technique based on hue value detection was used to measure the heat transfer coefficient on a trailing edge film-cooling model with slot ejection and an internal model with perforated blockage inserts. It was also used to determine the film effectiveness on the cutback trailing edge. They showed that the internal design geometry of the trailing edge and Reynolds numbers could affect heat transfer in an internal model with perforated blockage inserts. The wider entrance channel and a sloped land near the ejection slots provided low heat transfer coefficients in the internal as well as external model but gave higher film-cooling effectiveness from slot ejection.

Bunker and Bailey (2001) studied the film cooling discharge coefficient measurements in a turbulated passage with internal cross-flow. They investigated the effect of turbulator rib placement on film hole discharge coefficient. In the study, a square passage having a hydraulic diameter of 1.27 cm was used to feed a single angled film jet. The film hole angle to the surface was 30 deg and the hole length-to-diameter ratio is 4. Turbulators were placed in one of three positions: upstream of film hole inlet, downstream of film hole inlet, and with the film hole inlet centered between turbulators. For each case 90 deg turbulators with a passage blockage of 15 percent and a pitch to height ratio of 10 were used. Tests were run varying film hole-to-crossflow orientation as 30, 90, and 180 deg, pressure ratio from 1.02 to 1.8, and channel cross-flow velocity from Mach 0 to 0.3. Film hole flow was captured in a static plenum with no external cross-flow. They concluded that the alignment of the film hole entry with respect to the turbulator had a substantial effect on the resulting discharge coefficients. Depending on the relative alignment and flow direction discharge coefficients could be increased or decreased 5–20 percent from the nonturbulated case, and in the worst instance experience a decrease of as much as 50 percent.

Thole et al. (1997) studied the effect of different velocities in a coflowing channel at the film cooling hole entrance. Flows on both sides of the cooling hole (entrance and exit) were parallel and in the same direction. With the blowing ratio and the main stream velocity at the hole exit remaining fixed, only the flow velocity in the channel at the hole entrance was varied. Flow field measurements were made inside the hole, at the hole inlet and exit, and in the near-hole region where the jet interacted with the cross flow at the hole exit. They concluded that for entrance cross flow Mach numbers of Mac = 0 and 0.5, a separation region occurred on the leeward and windward side of the cooling hole entrances, respectively. As a result of this separation region, the cooling jet exited in a skewed manner with very high turbulence levels.

Thole et al. (1998) compared the flow field measurements for three different single scaled-up hole geometries, all at a blowing ratio and density ratio of unity. The hole geometries included a round hole, a hole with a laterally expanded exit, and a hole with a forward-laterally expanded exit. In addition to the flow field measurements for expanded cooling hole geometries, the testing facility used for these measurements was also in that both the external mainstream Mach number ($Ma\infty = 0.25$) and internal coolant supply Mach number ($Mac = 0.3$) were nearly matched. They concluded that by expanding the exit of the cooling holes, both the penetration of the cooling jet and the intense shear regions were significantly reduced relative to a round hole. Although the peak turbulence level for all three hole geometries was nominally the same, the source of that turbulence was different. The peak turbulence level for both expanded holes was located at the exit of the cooling hole resulting from the expansion angle being too large. The peak turbulence level for the round hole was located downstream of the hole exit where the velocity gradients were very large.

A three-dimensional Navier–Stokes simulation was performed by Heidmann et al. (2000) for a realistic film cooled turbine vane using the LeRC-HT code. The simulation included the flow regions inside the coolant plena and film cooling holes in addition to the external flow. The vane had both circular cross-sectional and shaped film cooling holes and was modeled using a multi-block grid, which accurately discretized the actual vane geometry including shaped holes. The simulation matched operating conditions for the planned experiment and assumed periodicity in the span-wise direction on the scale of one pitch of the film cooling hole pattern. Two computations were performed for different isothermal wall temperatures, allowing independent determination of heat transfer coefficients and film effectiveness values. The results indicated separate localized regions of high heat flux in the showerhead region due to low film effectiveness and high heat transfer coefficient values, but the shaped holes provided a reduction in heat flux through both parameters. Hole exit data indicated rather simple skewed profiles for the round holes, but complex profiles for the shaped holes with mass fluxes skewed strongly toward their leading edges. A hybrid approach based on the modeling of the near film-cooling hole flow, tuned using experimental data, while computing directly the flow field in the blade-to-blade passage has been proposed by Bernsdorf et al. (2006) and Burdet et al. (2007). A new injection film-cooling model was established to simulate film-cooled turbine blades without needing explicit meshing inside the holes and the plenum chamber. The experimental facility simulated the film cooling row flow field on the pressure side of a turbine blade. Engine representative nondimensionals were achieved, providing a faithful model at larger scale. The three dimensional velocities were recorded using nonintrusive PIV and seeding was provided for both air streams. Two different cylindrical hole geometries were studied, with different angles. Blowing ratio was varied over a range to simulate pressure side film cooling. The three dimensional flow structures were revealed.

Gao et al. (2007) studied four kinds of hole film cooling hole configurations: cylindrical, laterally-diffused (fan-shape), forward-diffused (laid-back), and laterally- and forward-diffused (laid-back fan-shape). The film cooling effectiveness distribution produced by the various hole configuration is also conceptually indicated in Figure. 2.1 with the dashed lines. The film cooling effectiveness on the surface of a high pressure turbine blade was measured using the pressure sensitive paint (PSP) technique. The coolant was only injected to either the pressure side or suction side of the blade at five average blowing ratios ranging from 0.4 to 1.5. Results revealed that the tip leakage vortices and end-wall vortices sweep the coolant film on the suction side to the mid-span region.

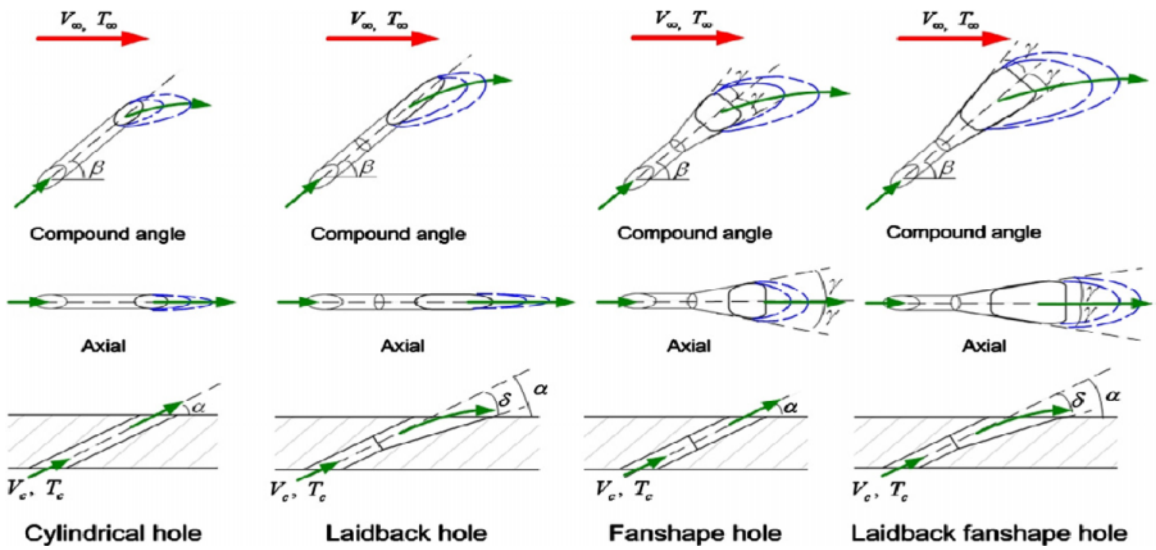

Figure 2.1: Film cooling hole configurations (Gao et al., 2007).

The work of Renze et al. (2008) investigated the impact of the velocity and density ratio on the turbulent mixing process in gas turbine blade film cooling, using large-eddy simulations. A cooling fluid was injected through an inclined pipe at $\alpha = 30^0$ into a turbulent boundary layer at a free-stream Reynolds number of 400,000. The governing equations comprised the Navier–Stokes equations plus additional transport equations for several species to simulate a non-reacting gas mixture. That is, gases of different densities were effused into an air cross-flow at a constant temperature. An efficient large-eddy simulation method for low subsonic flows based on an implicit dual time-stepping scheme combined with low Mach number preconditioning was applied. The results evidenced the dynamics of the flow field in the vicinity of the jet hole, i.e., the recirculation region and the inclination of the shear layers, to be mainly determined by the velocity ratio. However, evaluating the cooling efficiency downstream of the jet hole the mass flux ratio proved to be the dominant similarity parameter, i.e., the density ratio of the jet and cross flow fluid had to be considered.

Ahn et al. (2007) had studied the effect of rotation on detailed film cooling effectiveness distributions in the leading edge region of a gas turbine blade with three showerhead rows of radial-angle holes using the Pressure Sensitive Paint (PSP) technique. Tests were conducted on the first-stage rotor blade of a three-stage axial turbine at three rotational speeds. The effect of the blowing ratio was also studied. The Reynolds number based on the axial chord length and the exit velocity was 200,000 and the total to exit pressure ratio was 1.12 for the first-stage rotor blade. The corresponding rotor blade inlet and exit Mach number was 0.1 and 0.3, respectively. The film cooling effectiveness distributions were presented along with the discussions on the influences of rotational speed, blowing ratio, and vortices around the leading edge region.

There were 15 film cooling holes in three rows; pressure side, suction side and center row as shown in Figure 2.2. Each row had five film cooling holes and the holes along the central (y) axis were equally shared by the pressure and suction sides, as shown. The distance (p) between the film cooling holes was 9.5 mm in the span-wise direction and the film hole diameter (d) was 1.19 mm (p/d = 5.97). Figure 2.3 illustrates the flow path inside and outside the blade for (a) positive off-design, (b) design, and (c) negative off-design conditions. The results showed that different rotation speeds significantly change the film cooling traces with the average film cooling effectiveness in the leading edge region increasing with blowing ratio.

Figure 2.2: Modified film cooling blade with three rows of radial-angle holes (Ahn et al., 2007).

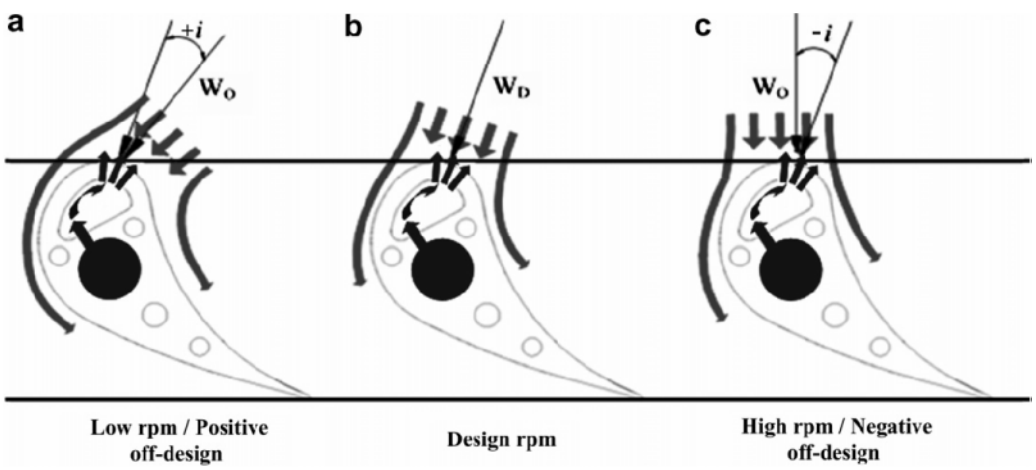

Figure 2.3: Flow paths inside and outside the blade for (a) positive off-design, (b) design, and (c) negative off-design conditions(Ahn et al., 2007).

The suitability of three different two-equation turbulence models; k-ε model, the k-ω model and the shear stress transport k- ω model, in predicting film cooling effectiveness on a rotating blade was investigated by Tao et al. (2009). To fulfill this target, both numerical simulation and the experimental investigation were carried out for a rotating blade having a flat test surface with a 4 mm diameter straight circular cooling hole in 30^0 inclined injection. The blade rotated at five different speeds of 0, 300, 500, 800 and 1000 rpm. They showed that the rotating speed was the most critical parameter influencing the film cooling effectiveness distributions and the pressure surface could be remarkably different from the suction surface. Further, among the three turbulence models, the standard k- ε model gave the poorest prediction.

2.2 Effect of Heat Transfer and Film Cooling on the Blade

Heat transfer coefficient and film cooling effectiveness on a gas turbine blade tip were measured by Kwak and Han (2003a) using hue detection based transient liquid crystals technique. Tests were performed on a five-bladed linear cascade with blow-down facility. The Reynolds number based on cascade exit velocity and axial chord length was 1.1×10^6 and the total turning angle of the blade was 97.7°. The overall pressure ratio was 1.2 and the inlet and exit Mach numbers were 0.25 and 0.59, respectively. The turbulence intensity level at the cascade inlet was 9.7%. The blade model was equipped with a single row of film cooling holes at both the tip portion along the camber line and near the tip region of the pressure side. All measurements were made at the three different tip gap clearances of 1.0%, 1.5%, and 2.5% of blade span and the three blowing ratios of 0.5, 1, and 2. Results showed that, in general, heat transfer coefficient and film effectiveness increased with increasing tip gap clearance. As blowing ratio increased, heat transfer coefficient decreased, while film effectiveness increased. Results also showed that adding pressure side coolant injection would further decrease the blade tip heat transfer coefficient but increase film-cooling effectiveness.

Wright et al. (2008) made heat transfer analysis on rotating cooling channel that was used to model an internal cooling passage near the trailing edge of a gas turbine blade. By varying the rotational speed of the channel, the rotation number and buoyancy parameter ranged from 0 to 1.0 and 0 to 3.5, respectively. Significant variation of the heat transfer coefficients has been found in both the span-wise and stream-wise directions. Span-wise variation was the result of the wedge-shaped design, and stream-wise variation was the result of the sharp entrance into the channel and the 180 deg turn at the outlet of the channel. With the channel rotating at 135° with respect to the direction of rotation, the heat transfer coefficients were enhanced on every surface of the channel.

Experimental and numerical investigations of convective heat transfer on the first-stage blade tip surface for a geometry typical of large power generation turbines had been performed by Bunker et al. (2000) and Ameri and Bunker (2000). The simulations in this study were performed using a multiblock computer code called LeRC-HT which is a general purpose flow solver designed for simulations of flows in complicated geometries. The code solves the full compressible Reynolds-averaged, Navier– Stokes equations using a multistage Runge–Kutta based multigrid method. It uses the finite volume method to discretize the equations. A stationary blade cascade experiment was run consisting of three airfoils, the center airfoil having a variable tip gap clearance. The airfoils were made according to the aerodynamic tip section of a high-pressure turbine blade with inlet Mach number of 0.30; exit Mach number of 0.75, pressure ratio of 1.45, exit Reynolds number based on axial chord of 2.57×10^6, and total turning of about 110 deg. A hue detection based liquid crystal method was used to obtain the detailed heat transfer coefficient distribution on the blade tip surface for flat, smooth tip surfaces with both sharp and rounded edges. Good comparison with the experimental measured distribution was claimed through accurate modeling of the most important features of the blade passage and heating arrangement as well as the details of experimental rig likely to affect the tip heat transfer. A sharp edge and a rounded edge tip were considered and the results of the rounded edge tip agreed better with the experimental data.

Ameri and Rigby (1999) had simulated the blade tip with cooling holes to predict the convective heat transfer coefficient distribution. The purpose of this examination was to assess the ability of a three-dimensional Reynolds-averaged Navier-Stokes solver to predict the rate of tip heat transfer and the distribution of cooling effectiveness. To this end, the simulation of tip clearance flow with blowing of Kim and Metzger was used. Numerical flow visualization showed that the uniformity of wetting of the surface by the film cooling jet was helped by the reverse flow due to edge separation of the main flow.

Heat transfer coefficient and static pressure distributions were experimentally investigated on a gas turbine blade tip in a five-bladed stationary linear cascade by Azad et al. (2000). The blade was a two-dimensional model of a first-stage gas turbine rotor blade with a blade tip profile of GE-E3 aircraft gas turbine engine rotor blade. The flow condition in the test cascade corresponded to an overall pressure ratio of 1.32 and exit Reynolds number based on axial chord of 1.1×10^6. The middle 3-blade had a variable tip gap clearance. All measurements were made at three different tip gap clearances of about 1, 1.5, and 2.5 percent of the blade span. Heat transfer measurements were also made at two different turbulence intensity levels of 6.1 and 9.7 percent at the cascade inlet. Static pressure measurements were made in the mid span and the near-tip regions as well as on the shroud surface, opposite the blade tip surface. Detailed heat transfer coefficient distributions on the plane tip surface were measured using a transient liquid crystal technique. Results showed various regions of high and low heat transfer coefficient on the tip surface. Tip clearance had a significant influence on local tip heat transfer coefficient distribution. Heat transfer coefficient also increased about 15–20 percent along the leakage flow path at higher turbulence intensity level of 9.7 over 6.1 percent.

Azad et al. (2002) investigated the effect of squealer tip geometry arrangement on heat transfer coefficient and static pressure distributions on a gas turbine blade tip in a five-bladed stationary linear cascade. A transient liquid crystal technique was used to obtain detailed heat transfer coefficient distribution. The test blade was a linear model of a tip section of the GE-E3 high-pressure turbine first stage rotor blade. Six tip