Effect Of Pressure Gradient And Wake On Endwall Film Cooling Effectiveness

Sylvette Rodriguez
University of Central Florida

Find similar works at: https://stars.library.ucf.edu/etd
University of Central Florida Libraries http://library.ucf.edu

This Doctoral Dissertation (Open Access) is brought to you for free and open access by STARS. It has been accepted for inclusion in Electronic Theses and Dissertations, 2004-2019 by an authorized administrator of STARS. For more information, please contact STARS@ucf.edu.

STARS Citation
https://stars.library.ucf.edu/etd/3703
EFFECT OF PRESSURE GRADIENT AND WAKE ON ENDWALL FILM COOLING EFFECTIVENESS

by

SYLVETTE RODRIGUEZ
B.S. University of Puerto Rico, 2000
M.S. University of Central Florida, 2005

A dissertation submitted in partial fulfillment of the requirements for the degree of Doctor of Philosophy in the Department of Mechanical, Materials, and Aerospace Engineering in the College of Engineering and Computer Science at the University of Central Florida Orlando, Florida

Fall Term
2008

Major Professor: Jayanta Kapat
ABSTRACT

Endwall film cooling is a necessity in modern gas turbines for safe and reliable operation. Performance of endwall film cooling is strongly influenced by the hot gas flow field, among other factors. For example, aerodynamic design determines secondary flow vortices such as passage vortices and corner vortices in the endwall region. Moreover blockage presented by the leading edge of the airfoil subjects the incoming flow to a stagnating pressure gradient leading to roll-up of the approaching boundary layer and horseshoe vortices. In addition, for a number of heavy frame power generation gas turbines that use cannular combustors, the hot and turbulent gases exiting from the combustor are delivered to the first stage vane through transition ducts. Wakes induced by walls separating adjacent transition ducts located upstream of first row vanes also influence the entering main gas flow field. Furthermore, as hot gas enters vane passages, it accelerates around the vane airfoils. This flow acceleration causes significant streamline curvature and impacts lateral spreading endwall coolant films. Thus endwall flow field, especially those in utility gas turbines with cannular combustors, is quite complicated in the presence of vortices, wakes and strong favorable pressure gradient with resulting flow acceleration. These flow features can seriously impact film cooling performance and make difficult the prediction of film cooling in endwall.

This study investigates endwall film cooling under the influence of pressure gradient effects due to stagnation region of an axisymmetric airfoil and in mainstream favorable pressure gradient. It also investigates the impact of wake on endwall film cooling near the stagnation region of an airfoil. The investigation consists of experimental testing and numerical simulation.
Endwall film cooling effectiveness is investigated near the stagnation region on an airfoil by placing an axisymmetric airfoil downstream of a single row of inclined cylindrical holes. The holes are inclined at 35° with a length-to-diameter ratio of 7.5 and pitch-to-diameter ratio of 3. The ratio of leading edge radius to hole diameter and the ratio of maximum airfoil thickness to hole diameter are 6 and 20 respectively. The distance of the leading edge of the airfoil is varied along the streamwise direction to simulate the different film cooling rows preceding the leading edge of the airfoil. Wake effects are induced by placing a rectangular plate upstream of the injection point where the ratio of plate thickness to hole diameter is 6.4, and its distance is also varied to investigate the impact of strong and mild wake on endwall film cooling effectiveness. Blowing ratio ranged from 0.5 to 1.5.

Film cooling effectiveness is also investigated under the presence of mainstream pressure gradient with converging main flow streamlines. The streamwise pressure distribution is attained by placing side inserts into the mainstream. The results are presented for five holes of staggered inclined cylindrical holes. The inclination angle is 30° and the tests were conducted at two Reynolds number, 5000 and 8000.

Numerical analysis is employed to aid the understanding of the mainstream and coolant flow interaction. The solution of the computational domain is performed using FLUENT software package from Fluent, Inc. The use of second order schemes were used in this study to provide the highest accuracy available. This study employed the Realizable $\kappa$-$\varepsilon$ model with enhance wall treatment for all its cases.
Endwall temperature distribution is measured using Temperature Sensitive Paint (TSP) technique and film cooling effectiveness is calculated from the measurements and compared against numerical predictions.

Results show that the characteristics of average film effectiveness near the stagnation region do not change drastically. However, as the blowing ratio is increased jet to jet interaction is enhanced due to higher jet spreading resulting in higher jet coverage. In the presence of wake, mixing of the jet with the mainstream is enhanced particularly for low M. The velocity deficit created by the wake forms a pair of vortices offset from the wake centerline. These vortices lift the jet off the wall promoting the interaction of the jet with the mainstream resulting in a lower effectiveness. The jet interaction with the mainstream causes the jet to lose its cooling capabilities more rapidly which leads to a more sudden decay in film effectiveness. When film is discharged into accelerating main flow with converging streamlines, row-to-row coolant flow rate is not uniform leading to varying blowing ratios and cooling performance. Jet to jet interaction is reduced and jet lift off is observed for rows with high blowing ratio resulting in lower effectiveness.
To my family, thank you for being a pillar of strength for me.
ACKNOWLEDGMENTS

This work has been possible by the help and cooperation of many. To my advisor, Dr. Jayanta Kapat, thank you for your support, dedication, and for all that I have learned from you. Your passion and continued desire to rise above inspires many. I also thank my thesis committee, Dr. Gordon, Dr Guha, Dr. Basu and in particular to Dr. Kassab, thank you for your time and continuous support throughout the years.

It has been a privilege to work alongside the members of the investigation group at the Center for Advanced Turbine and Energy Research (CATER): Quan, Vaidy, Chris, Humberto, Anil, Stephanie and others. Your motivation and sense of teamwork is admirable.

To all my professors who contributed to my professional growth. To my friends, colleagues, and to all who in one way or another have supported me and have taken part in this dissertation, my sincerest gratitude.

There are no words that could possibly express the appreciation that I have for my family. To my husband, Luis F., thank you for companionship through my late nights, for your commitment to me in this journey and your continued words of comfort. To my parents, my brother Benny and sister Mariel, whom have inspired me to supersede with your examples and have taught me the value of education and the importance of following after my dreams.

Above all, my deepest gratitude is for my Lord Jesus, whom without any of this would have meaning. May my whole life always bring glory and honor to Your name.
TABLE OF CONTENTS

LIST OF FIGURES ................................................................................................................................. xi

LIST OF TABLES ...................................................................................................................................... xvi

NOMENCLATURE ..................................................................................................................................... xvii

CHAPTER 1: INTRODUCTION ............................................................................................................ 1

CHAPTER 2: LITERATURE REVIEW .................................................................................................. 8

Experimental Studies ......................................................................................................................... 8
Computational Studies ....................................................................................................................... 14

CHAPTER 3: METHODOLOGY .......................................................................................................... 19

Experimental Methodology ................................................................................................................ 19
BFC Wind Tunnel Details .................................................................................................................. 19
IFC Wind Tunnel Details .................................................................................................................... 27
Temperature Sensitive Paint .............................................................................................................. 33
Computational Methodology .............................................................................................................. 37
Computational Model ......................................................................................................................... 37
Grid and Mesh Generation ................................................................................................................ 39
Solution and Discretization .................................................................................................................. 39
Turbulence Model ............................................................................................................................... 40
Near-Wall Treatments ......................................................................................................................... 41
LIST OF FIGURES

Figure 1: Gas temperature compared to allowable metal temperature ....................... 1
Figure 2: Damage to Nozzle Guide Vanes due to Excessive High Temperatures .......... 1
Figure 3: Development of Cooling Methods Techniques vs. Inlet Temperature ............ 2
Figure 4: Internal Cooling Passages on an Airfoil ................................................... 3
Figure 5: Different Film Cooling Methods (Eckert) ................................................. 3
Figure 6: Recovery Temperature vs. Adiabatic wall Temperature ............................ 4
Figure 7: Flow Model Proposed by Langston [2] ..................................................... 5
Figure 8: Schematic of the Endwall Boundary Layers (Denton and Cumpsty) ...... 6
Figure 9: MS7001FA Gas Turbine and Representation of Region under Investigation .... 7
Figure 10: Schematic of the near hole flow field [7] ............................................... 10
Figure 11: BFC Test Rig Schematic ........................................................................ 20
Figure 12: Location of thermocouples in BFC rig ................................................. 21
Figure 13: Scanivalve Calibration Curve ............................................................... 22
Figure 14: Flow meters ......................................................................................... 23
Figure 15: Acrylic test coupon ............................................................................. 24
Figure 16: Leading Edge radius to Film Cooling Hole Pitch Ratio ......................... 25
Figure 17: Test Setup of Airfoil with Wake plate .................................................. 25
Figure 18: Test Configurations for study on BFC ................................................. 26
Figure 19: Vertical section of the IFC tunnel ......................................................... 27
Figure 20: Horizontal section of the IFC tunnel .................................................... 28
Figure 21: Test Section Setup ................................................................. 31
Figure 22: Test Coupon ........................................................................... 32
Figure 23: Alignment of inserts with film hole exits .................................. 33
Figure 24: Emission spectrum of TSP (Liu, 2006) ...................................... 34
Figure 25: Typical TSP setup and instrumentation (Liu, 2006) ................... 35
Figure 26: CCD camera and light source .................................................. 36
Figure 27: Spectrum of LED source (Liu, 2006) ......................................... 36
Figure 28: Computational Domain and Boundary Conditions Type (Configuration 0-a) 38
Figure 29: Inlet Total Pressure Profile ....................................................... 38
Figure 30: Grid Independence Check ....................................................... 43
Figure 31: Mach number distribution along the film hole rows .................. 45
Figure 32: Normal image of TSP ............................................................... 48
Figure 33: Reference Image ................................................................. 49
Figure 34: Image of TSP showing coolant flow .......................................... 50
Figure 35: TSP Calibration Curve ......................................................... 50
Figure 36: Raw Temperature image from TSP ............................................ 51
Figure 37: Results of averaging effectiveness ........................................... 52
Figure 38: Uncertainty distribution for film cooling effectiveness [29] ........ 53
Figure 39: Uncertainty in $C_D$ measurement [29] .................................... 53
Figure 40: FLUENT Local Temperature Data extracted for calculating effectiveness .... 55
Figure 41: Graphical User Interface to process Computational Data .......... 56
Figure 42: Process flowchart of Data Reduction of Computational Data ....... 57
Figure 43: Configuration 0-a CFD Grid ................................................................. 60
Figure 44: CFD Domain and Boundary Conditions Type for Configuration 0-a........ 60
Figure 45: Data Comparison for M=0.5 ................................................................. 61
Figure 46: Data Comparison for M=0.75 .............................................................. 62
Figure 47: Comparison of Turbulence Models against Experimental Data .......... 63
Figure 48: Validation of Configuration 0-a ............................................................ 64
Figure 49: Average Effectiveness for Configuration 0-a ......................................... 65
Figure 50: Local Temperature Distribution for Configuration 0-a ......................... 65
Figure 51: Local Temperature Distribution for Configuration 0-a (Exp vs. CFD) .... 65
Figure 52: Streamline Plot for Configuration 0-a (M=0.5) .................................... 66
Figure 53: Streamline Plot for Configuration 0-a (M=1.5) .................................... 67
Figure 54: Test Configurations for study on stagnation region effects ..................... 68
Figure 55: Stagnation Region Effects CFD Grid ..................................................... 72
Figure 56: CFD Domain and Boundary Conditions Type for Configuration I and III-a. 73
Figure 57: Average Film Cooling Effectiveness for Configuration I-a ................. 74
Figure 58: Local Film Cooling Effectiveness for Configuration I-a (Exp. vs. CFD) .... 74
Figure 59: Average Film Cooling Effectiveness for Configuration I-b ................. 75
Figure 60: Local Film Cooling Effectiveness for Configuration I-b (Experimental) .... 75
Figure 61: Average Film Cooling Effectiveness for Configuration I-c ................. 76
Figure 62: Local Film Cooling Effectiveness for Configuration I-c (Exp. vs. CFD) .... 76
Figure 63: Average Film Cooling Effectiveness (Configuration 0, I, and III-a) ......... 77
Figure 64: Streamline Plot for Configuration I-a (M=0.5 and 1.5) ......................... 77
Figure 87: Local Film Cooling Effectiveness for Configuration III-b ......................... 99
Figure 88: Local Film Cooling Effectiveness for Configuration 1-c............................ 99
Figure 89: Local Film Cooling Effectiveness for Configuration II-c.......................... 99
Figure 90: Local Film Cooling Effectiveness for Configuration III-c......................... 100
Figure 91: Streamline plots for Configuration I-b ...................................................... 100
Figure 92: Streamline plots for Configuration III-b .................................................... 100
Figure 93: Streamline plots for Configuration I-c ...................................................... 101
Figure 94: Streamline plots for Configuration III-c .................................................... 101
Figure 95: Film Coverage ......................................................................................... 102
Figure 96: Pressure Gradient CFD Grid ................................................................. 104
Figure 97: Pressure Gradient CFD Domain and Boundary Conditions Type ............. 105
Figure 98: Average Film Cooling Effectiveness (Re=5000 and 8000) ....................... 106
Figure 99: Temperature distribution at Re=5000 ..................................................... 107
Figure 100: Temperature distribution at Re=8000 ..................................................... 107
Figure 101: Streamline Plots (Re = 5000 and 8000) ............................................... 108
Figure 102: Local Blowing Ratios for M = 0.5 ......................................................... 109
Figure 103: Local Blowing Ratios for M = 1.5 ......................................................... 110
Figure 104: Row by Row Blowing Ratio Variation (Re = 5000) ............................... 111
Figure 105: Row by Row Blowing Ratio Variation (Re = 8000) ............................... 111
LIST OF TABLES

Table 1: Test Matrix for BFC Tunnel ................................................................. 26
Table 2: Baseline Test Experimental Measurements ........................................... 59
Table 3: Test Configurations for study on stagnation region effects .................. 68
Table 4: Experimental Measurements for Configuration I-a ............................... 69
Table 5: Experimental Measurements for Configuration II-a ............................ 70
Table 6: Experimental Measurements for Configuration III-a ........................... 71
Table 7: Test Matrix for BFC Tunnel ................................................................. 79
Table 8: Experimental Measurements for Configuration 0-b ............................... 81
Table 9: Experimental Measurements for Configuration 0-c ............................... 82
Table 10: Test Matrix for BFC Tunnel ............................................................... 88
Table 11: Experimental Measurements for Configuration I-b ............................. 89
Table 12: Experimental Measurements for Configuration III-b ........................ 90
Table 13: Experimental Measurements for Configuration I-c ............................. 91
Table 14: Experimental Measurements for Configuration III-c .......................... 92
Table 15: Boundary Conditions, Re= 5000 ..................................................... 103
Table 16: Boundary Conditions, Re= 8000 .................................................... 103
NOMENCLATURE

A   Area
a   Actual
avg  Average
aw  Adiabatic wall
c  Coolant
C  Specific Heat
CD  Discharge Coefficient
D  diameter of hole inlet
f  Film (Coolant flow)
FCR  Film Coverage Ratio, FCR=
h  holes
I  Momentum Flux ratio
K  Acceleration Parameter, \( K = \frac{\nu}{U^2} \left( \frac{\partial U}{\partial x} \right) \)
L  Length of hole
M  Blowing ratio
m  Main flow
\( \cdot m \)  Mass Flow Rate
P  Pressure
PI  Pitch
Pr Prandtl Number
r Recovery
RKE Realizable $\kappa$-$\epsilon$ model
s Pitch
T Temperature
th theoretical
Tu Turbulence Intensity
U,V Velocity
w Wall
x Distance downstream of center of the hole

Greek Symbols

0 Stagnation value
$\infty$ Mainstream condition
$\infty$ Hole Inclination Angle
$\delta$ Boundary Layer Thickness
$\eta$ Film cooling effectiveness
$\rho$ Density
CHAPTER 1: INTRODUCTION

The gas turbine industry is continuously under demand to attain higher performance, longer service life and lower emissions. The increasing demand to optimize gas turbines subjects hot section components to more extreme operating conditions. The temperature of the gases entering the turbine section is much greater than the allowable metal temperatures of the alloys used in the turbine airfoils and endwall. Such high temperatures result on cracks, material deformation, corrosion and fracture, as shown on Figure 2.

![Figure 1: Gas temperature compared to allowable metal temperature](Cardwell, N. D., 2005)

![Figure 2: Damage to Nozzle Guide Vanes due to Excessive High Temperatures]
The continuous effort to increase turbine efficiency has promoted the development and sophistication of new material, thermal barriers and cooling methods. Different cooling methods have been implemented to protect metal surfaces from hot gases; among them are film impingement, ablation, transpiration and film cooling.

![Figure 3: Development of Cooling Methods Techniques vs. Inlet Temperature (Clifford, 1985, AGARD CP 390; collected in Laskshminarayana, 1996)](image)

In gas turbines air is drawn from the compressor and ejected via rows of holes located at discrete locations on the airfoil and endwall to cool the surfaces. This process is called film cooling. It consists of introducing a secondary flow into the boundary layer of the mainflow. Film cooling not only protects the surface at the point of injection but also region downstream of the injection point. The ejected coolant interacts with the external flow near the endwall and generates aerodynamic and thermodynamic losses in the process. This reduces turbine stage efficiency and the consumption of cooling air, which is detrimental to the overall cycle efficiency. Over the recent years, existing cooling techniques are being optimized and new
endwall film cooling configurations are continuously explored. The data presented in Figure 5, shows that film cooling is the most effective cooling methods among those developed in the recent years.

![Figure 4: Internal Cooling Passages on an Airfoil (Energy Efficient Engine (E3) NASA Report)](image)

Endwall film cooling is determined among others by the inlet temperature distribution, flow field and the state and thickness of the endwall boundary layer. The condition of the flow entering the first stage vane is greatly affected by the hot and turbulent gases exiting from the combustor. In order for this cooling technique to be effective, film cooling must provide acceptable metal surface temperatures and reasonable temperature distribution over the endwall to limit thermal stress as a result of large temperature gradients.
Goldstein [1] described that the convective heat transfer associated with film cooling is determined by using a film heat transfer coefficient \( h_f \), an adiabatic wall or film temperature \( T_{aw} \), where \( T_w \) is the local wall temperature:

\[
q = h_f \cdot (T_w - T_{aw})
\]  

(Equation 1)

It is convenient to define a dimensionless adiabatic wall temperature, \( \eta \), called film cooling effectiveness. This dimensionless parameter is given by:

\[
\eta = \frac{T_r - T_{aw}}{T_r - T_c}
\]  

(Equation 2)

where \( T_r \) is the mainstream recovery temperature of the hot gas flow, and \( T_c \) is the stagnation coolant temperature at the point of injection. The recovery temperature of the gas is calculated by:

\[
T_r = T_\infty + \text{Pr}^{1/3} \cdot \frac{V^{1/2}}{2 \cdot C_p}
\]  

(Equation 3)

\( T_r \) is the surface temperature of an un-cooled insulated surface as shown in Figure 6.

Figure 6: Recovery Temperature vs. Adiabatic wall Temperature
Thus, the film cooling effectiveness varies from unity at the point of injection to zero where the adiabatic wall temperature approaches the mainstream temperature due to mixing between the coolant flow and the mainstream.

The flow near the endwall into which the coolant is being ejected can be inherently three-dimensional. Due to the blockage presented by the leading edge of the airfoil, the incoming flow is subjected to a stagnating pressure gradient. This causes the flow to undergo three-dimensional separation and to roll up into a horseshoe vortex. Secondary flows make difficult the prediction of the behavior of film cooling in the stagnation region. Moreover, the turning of the mainstream flow within the vane passage together with airfoil curvature may produce a vane-to-vane pressure gradient that generates a transverse component of flow within the endwall boundary layer. The ejected coolant interacts with this three-dimensional flow. The flow can influence coolant trajectories and to a smaller extend, the ejection of coolant has the potential of influencing the three-dimensional flow.

Figure 7: Flow Model Proposed by Langston [2]
In most heavy frame power generation turbines, the hot and turbulent gases exiting from the combustor are delivered to the first stage vane through transition ducts. Wakes induced by the wall separating adjacent transition ducts located upstream of first row vanes, could also influence the entering main gas flow field. The incoming flow field condition makes difficult the prediction of the behavior of endwall film cooling near in the stagnation region as the film approaches the turbine vane. Although thorough studies have been carried out on endwall film cooling, the effect of the transition ducts on endwall film cooling near stagnation region is not fully investigated.
The optimization of a cooling system has to be weighed against the increase in cycle efficiency that can be achieved through higher turbine entry temperatures. Up to the present, there are no correlations or experimental data available that characterize endwall film cooling effectiveness near an airfoil’s stagnation region under the influence of wake induced by the transition ducts and the pressure gradient induced by the incoming flow condition. This dissertation presents the physics of flow mechanics responsible for determining downstream heat transfer in the nearby region of the stagnation point of an axisymmetric airfoil, and under the influence of pressure gradient and wake.
CHAPTER 2: LITERATURE REVIEW

Experimental Studies

Film cooling is a widely used method of cooling different components of aviation gas turbines and power generation units. Film cooling enables turbines to operate at higher temperatures than what the substrate super alloy can withstand. For decades, investigations on film cooling techniques have been carried out for improving its performance to meet the ever increasing turbine operating temperatures. In most gas turbine applications, the structural loading at which its components are subjected does not allows a secondary flow to be injected through slots; instead discrete holes are used to inject coolant into the mainstream.

Film cooling effectiveness depends on blowing ratio (M), momentum flux ratio (I), mainstream turbulence intensity (Tu), length-to-diameter ratio (L/D), pitch-to-diameter ratio (s/D), hole inclination angle (α), crossflow angle, displacement thickness of the mainstream boundary layer -to-diameter ratio (δ/D), density ratio, and hole geometry. In the past years, there have been several studies on film endwall film cooling where these important parameters have been investigated. The blowing ratio and momentum flux ratio are defined as:

\[
M = \frac{\frac{m}{A}}{\frac{m}{A}} = \frac{\frac{(\rho \cdot V \cdot A)_c}{A_c}}{\frac{(\rho \cdot V \cdot A)_m}{A_m}} = \frac{\rho_c \cdot V_c}{\rho_m \cdot V_m} \quad \text{(Equation 4)}
\]
and

\[ I = \frac{\rho_c \cdot \left( \frac{V^2}{\rho} \right)_c}{\rho_n \cdot \left( \frac{V^2}{\rho} \right)_m} \]  

\text{(Equation 5)}

Nicklas [3] studied the effect of Mach number and blowing ratio on endwall heat transfer and film-cooling effectiveness. Three rows of holes and a slot were investigated and found that the interaction of coolant air and the secondary flow field greatly influenced heat transfer and film effectiveness. It was observed that a decrease in intensity of the horseshoe vortex near the endwall results in a reduction in heat transfer. Similarly, Zhang and Jaiswal [4] studied film cooling effectiveness on a flat turbine nozzle endwall. It was reported that for low blowing ratios, the secondary flows dominate over the momentum of the jet, resulting in a decrease in effectiveness. On the contrary, at high blowing rates, the jet momentum overcomes the flow field yielding to a higher effectiveness. According to Goldstein et al. [5], the low effectiveness observed at low blowing ratios can be attributed to the jet spreading angle. He observed that for low blowing ratios, the spreading of the jet does not vary significantly with the injection angle. However, at a higher blowing ratio (M=1), the spreading angle increases at an injection of 90º and decreases for blowing ratios greater than 1. This investigation was conducted on an adiabatic flat plate for a row of holes inclined at 35º and 90º. Similarly, Eriksen and Goldstein [6] studied the effect of blowing rates on heat transfer coefficient. They also found that for a 35º row of film holes, high blowing rates increase heat transfer coefficient up to 27% for a blowing ratio of two. This increase is attributed to the interaction between the jets and the mainstream, resulting on an increase in the levels of turbulence. From these findings it can be inferred that
the spreading of the jets as described by Goldstein [5] enhances heat transfer and promotes turbulence levels.

![Diagram of near hole flow field](image)

**Figure 10:** Schematic of the near hole flow field [7]

The exiting jet can produce a broad range of flow structures downstream of the injection point. The jet as it exits is highly three-dimensional and contains flow features that are important for the diffusion of the coolant fluid with the mainstream flow. At the point of injection the coolant jet bends due to the inertial force of the mainstream flow and immediately starts to mix. High blowing ratio can cause the coolant flow to lift off. The coolant flow penetrating into the boundary layer can significantly reduce film cooling effectiveness. As depicted in Figure 10, even at moderate blowing ratios a wake zone is formed in the lower part of the jet, downstream of the film hole. This lowers the penetration velocity of jet allowing the freestream fluid to penetrate more easily underneath of the core of the jet.

The difference of flow properties between the injected flow and mainstream as well as the hole geometry also define the flow structure of the jet as it is injected into the mainstream.

The difference of flow properties between the injected flow and mainstream as well as the hole geometry also define the flow structure of the jet as it is injected into the mainstream.
The effect of density ratio on film cooling effectiveness was studied by Sinha et al. [8] and Pedersen et al. [9]. Sinha et al. [8] study was carried out for a single row of cylindrical holes inclined at 35°. The density ratio ranged from 1.2 to 2.0. It was found that a decreased in density ratio reduces significantly the spreading of the film cooling jet resulting in a reduction of the laterally averaged effectiveness. Pedersen et al. [9] carried out an investigation for a single row of holes inclined at 35°. Helium, carbon dioxide and refrigerant F-12 were used as coolant flow injected into the mainstream. Thus, the density ratio varied from 0.75 to 4.17. Similarly, this study reports that density ratio has a direct effect on film cooling effectiveness. This investigation reports that an increase in density ratio increases film cooling effectiveness. The influence of hole length-to-diameter ratio on film cooling with cylindrical holes was studied by Lutum and Johnson [10]. The study was conducted on a row of cylindrical holes inclined at 35° with varying length-to-diameter ratio (L/D = 1.75, 3.5, 5 and 7). It was determined that film cooling effectiveness decreased with decreasing L/D. This is attributed to (1) undeveloped character of the injected flow and (2) greater injection angle. Baldauf et al. [11] investigated the effect of hole spacing. Their study shows that for pitch-to-diameter ratio of five or larger, the lateral jet interaction is reduced significantly resulting on a decrease in effectiveness. For lower s/D effectiveness is completely dominated by jet interaction at a blowing rate equal to 1.7. The study results in a new correlation valid without any exception from the point of injection to the region downstream, for all combinations of flow and geometry parameters for a row of film holes in a flat plate.

Turbine inlet conditions in gas turbines generally consist of temperature and velocity profiles that vary in the spanwise and pitchwise directions resulting from combustor exit
conditions. Pressure gradients in turbines can be attributed to two reasons: the inherent pressure gradient from the turning of the flow and the pressure gradient resulting from the non-uniformity of the incoming flow along the radial span of the airfoil passage. As a consequence, the leading edge horseshoe vortex formed as the incoming boundary layer approached the stagnation region of the vane separates into a pressure and suction side horseshoe vortex legs. As described by Kost and Nicklas [12] the intensity of the horseshoe vortex is an important cause for the transport of coolant from the endwall to the main flow. In this study it was recommended that in order to prevent such an intensification of the horseshoe vortex, the distance from the injection point to the leading edge should be increased.

Ammari et al. [13] studied the influence of mainstream acceleration on the heat transfer coefficient for a single row of holes inclined at 35° in a flat plate. Two cases with two constant acceleration parameters were analyzed in addition to the zero pressure gradient baseline case. For the first case, K=1.96E-06 while for the second case K=5.0E-6. In addition two species were used as injected air, air and carbon dioxide yielding a density ratio of 1.0 and 1.52 respectively. The blowing ratio was also varied from 0.5 to 2.0. They found that heat transfer coefficient increased as the acceleration increased. Also, it was reported that in the presence of acceleration, the heat transfer coefficient was dependent of the density ratio, an increase in density ratio lead to a decrease in heat transfer coefficient. The pressure gradient was achieved by contouring the roof of the test section. In 1991, a similar film cooling effectiveness study was carried out by Teekaram et al. [14]. They report effectiveness for a single row of cylindrical holes on a flat plate inclined at 30°. Favorable and adverse pressure gradients are obtained by contouring the wall opposite to the injection location. The study reports that a strong favorable
pressure gradient increases effectiveness. Lower effectiveness is measured in the presence of adverse pressure gradient.

The literature contains some contradictory results about the influence of pressure gradient on film cooling effectiveness. Kruse [15] reported film cooling effectiveness measurements for a range of blowing ratios and pitch to diameter ratio. Three different injection angles were also studied, 10°, 45° and 90°. The pressure gradient was achieved by using a flexible wall to produce zero, favorable and adverse pressure gradient. The results of this paper show that pressure gradient has a significant effect on film cooling effectiveness most noticeable at low blowing ratios and small hole spacing. Adverse pressure gradient promotes jet lift off at locations close to the injection point but results in an increase in the effectiveness farther downstream. In the presence of favorable pressure gradient, the jet does not penetrate far into the mainstream. At the exit of jet, the jet is flattened out due to the contraction of the mainflow but the coolant does not mix well downstream of the injection point resulting in a lower film cooling effectiveness. Similarly to Hay et al. [16], Kruse observed that the boundary layer thickness seems not to have an effect on film cooling effectiveness. Similarly, Schmidt and Bogard [17] studied the effect of pressure gradient on film cooling effectiveness. They investigated a single row of cylindrical film cooling holes inclined at 35°. The top wall of the test section was contoured to produce the desired pressure gradient. The investigation reported that mainstream acceleration caused a faster decay in film cooling effectiveness, this is as a result of an increase in turbulence and hence an increase in the rate at which the jets mix with the mainstream. It was also reported a higher lateral averaged effectiveness near the injection region for all blowing ratios. The effect of heat transfer coefficient under the influence of pressure
gradient was investigated by Hay et al., [18] [16]. In these papers they also investigate the condition of the approaching boundary layer. The investigation was done for a single row of film cooling holes inclined at 35 and 90°. It was reported that the condition of the boundary layer has no significant effect on the heat transfer coefficient. In addition, for the same injection angles, a contoured test section roof produced the effect of mild adverse, mild favorable and strong favorable pressure gradient. It was reported that strong favorable pressure gradient relaminarizes the boundary layer, resulting in a drastically reduction in the heat transfer coefficient. It was concluded that the heat transfer coefficient is not sensitive to mild adverse pressure gradient.

The studies presented for pressure gradient studied the effect of flow acceleration around vane airfoils. This flow acceleration causes significant streamline curvature and impacts lateral spreading endwall coolant films. This study focuses on endwall film cooling, which will impact the spreading of the jet in the lateral direction.

**Computational Studies**

Researchers have also attempted to numerically predict film cooling performance. Over the last years, different turbulence models have been developed in order to more accurately predict the case of interest. Knost and Thole [19] presented results from a computational study for two endwall film cooling hole patterns of the first stage nozzle guide vane. Computations were performed for a pressure based incompressible flow for unstructured mesh. Reynolds Average Navier Stokes (RANS) equations as well as the energy and turbulent equation were
solved using a second order discretization. The two equation turbulence model used was RNG 
κ-ε model with non-equilibrium wall function. Their results show that the jet trajectory is highly
dependent on the local blowing ratio for the cooling holes. This study emphasizes the
importance of the presence of coolant leakage flow along the region where two turbine vanes are
mated. This coolant leakage flow provides cooling to the regions where the endwall film cooling
does not. Their findings are consistent with reported experimental data.

In 2000, Bernsdorf et al. [20] and Burdet et al. [7] reported experimental and computation
results for a single row of holes on a flat plate. The study simulated the effects of film cooling
row flow field on the pressure side of a turbine blade. In addition, two different cylindrical hole
injection angle were investigated. Blowing ratio was varied over a range typical of the film
cooling on a pressure side of a turbine blade. They used CFD code based on solving the
unsteady compressible RANS equation, MULT13. The solution method is based on an explicit,
finite volume, node based Ni-Lax-Wendroff time marching algorithm. This paper is relevant in
that it emphasizes the importance of accurately model the near hole region to capture the three
dimensional flow structure of the coolant jet. Another aspect of accurately defining the
numerical study is the wall treatment definition specified in the model. Walters and Leylek [21]
compared numerical results to experimental data for a single row of inclined holes. The length
to diameter ratio as well as the blowing ratio was varied. This study compared two layer wall
treatments versus wall functions. Their study generated unstructured mesh with wall treatment
values close to unity. The authors indicate that the combination of κ-ω with wall functions
seems to be the standard approach for such complex problems. However, this study reveals that
the use of two-layers wall treatment instead of wall functions significantly increases the
computational intensity of the simulation. From this investigation it was concluded that the two-layers model is necessary in order to resolve the recirculating flow beneath the exiting jet. Far field results from the two-layers model are almost equivalent to those predicted with wall functions.

Standard $\kappa$-$\epsilon$ turbulence model is widely found in literature as a turbulence model of choice for many researchers. Brittingham and Leylek [22] presented computational and experimental results for a single row of holes inclined at 35°. Although, the film hole geometry under investigation were variations of compound angle injection with shaped holes, the computational methodology employed in this study could be of benefit to this investigation. In this study they used multiblock, unstructured, pressure-correction code with multigrid and underrelaxation type convergence acceleration. The two-equation turbulence model was standard $\kappa$-$\epsilon$. Near wall treatments were also employed. Their results are in good agreement with the presented experimental data. Hassan and Yavuzkurt [23] recently published a paper comparing four different two-equation turbulence models predictions on film cooling performance. The models used are: standard k-e, RNG, realizable k-e and standard k-w, all available in FLUENT, same modeling tool proposed in this work. In all four models, enhanced wall treatments were used to resolve the flow near the solid boundaries. The computational study simulated a single row of holes inclined at 30°. The mainstream domain was meshed using a structured mesh whereas the other domain has a non-conformal (unstructured) mesh. This study concluded that standard k-e model had the most consistent performance and the best centerline effectiveness predictions among all turbulence models.
Other researchers have chosen $\kappa$-$\omega$ turbulence models as the model that better predicts film cooling performance. Adami et al. [24] discusses the aerodynamic interaction between film cooled blade and the periodic wake produced by a moving row of bars placed in a plane upstream of the cascade. The CFD results are compared against the available experimental data. They used spatial discretization based on a second order upwind TVD (Total Variation Diminishing) finite element volume approach for hybrid unstructured grid. The turbulence model employed was $\kappa$-$\omega$ by Wilcox. Their contribution is that the effect of wake does not change the global flow field structure of the vane, but it affects more appreciably the local features of the coolant mixing and separation beneath the ejecting holes. Kohli and Bogard [25] in a recent publication, studied the performance of film cooling using airfoil contouring. This study is a CFD based optimization study. For their simulation, a structured multiblock mesh was used and $\kappa$-$\omega$ turbulence model. Both airfoil and film hole were meshed using a O-H grid topology. The reported $y+$ values were of the order of one on no-slip boundaries.

Among the literature reviewed realizable $\kappa$-$\varepsilon$ turbulence model has been reported to be the best predictor of film cooling performance and endwall temperature distribution. Silieti et al. [26] investigated film cooling effectiveness for a fan shaped hole using three different two-equations turbulence models: realizable $\kappa$-$\varepsilon$, SST $\kappa$-$\omega$ and $\nu^2$-$f$ (V2F) models. Their study presents a multiblock numerical grid. The fan shaped holes were meshed using different mesh topologies: (1) hexahedral, (2) hybrid and (3) tetrahedral. Wall treatment values were in the order of one or less. Little effect on the grid topology was observed. However, RKE turbulence model best predicts centerline film cooling effectiveness among all other turbulence models investigated.
Walters and Leylek [21] similarly modeled film cooling problems using multiblock approach, unstructured mesh with a near wall treatment $y^+$ value of the order of one. Proper turbulence modeling is critical to accurately predict film cooling effectiveness. They reported that realizable $\kappa$-$\varepsilon$ model resulted in close agreement with the measured results. In this study, film cooling holes were located on an airfoil.

Silieti et al. [27] studied film cooling effectiveness in a 3D gas turbine endwall with one cylindrical hole inclined at $30^\circ$. Five different turbulence models were used: (1) standard $\kappa$-$\varepsilon$, (2) RNG $\kappa$-$\varepsilon$, (3) realizable $\kappa$-$\varepsilon$, (4) standard $\kappa$-$\omega$ and (5) SST $\kappa$-$\omega$ model. The computational work was set up using multiblock and all blocks in hexahedral mesh. Their findings were that RKE performed better than all other models. This turbulence model was in closer agreement with the predicted the surface temperature distribution.

Secondary flows typical in airfoil passages are the main contributors to heat transfer to the airfoil endwall. Due to the blockage presented by the leading edge of the airfoil, the incoming flow is subjected to a stagnating pressure gradient causing the flow to undergo three-dimensional separation forcing the boundary layer to reorganize into a horseshoe vortex. The airfoil leading edge region has grown in interest because it has been found that the horseshoe vortices increase local endwall heat transfer significantly. Although some investigations have been carried out to understand the complex three-dimensional flow in this region, the behavior of the film cooling on this portion of the endwall is not fully understood. In the following section attention will be given to the experimental set up, numerical approach and experimental and numerical results.
CHAPTER 3: METHODOLOGY

This investigation was a combination of experimental testing and computational analysis. Details of the test apparatus, data processing and computational analysis will be discussed in this section.

**Experimental Methodology**

Tests were conducted in the Basic Film Cooling (BFC) rig and Impingement Film Cooling (IFC) rig in the Engineering Field Lab facilities located on the main campus of the University of Central Florida. The study on the nearby region of an stagnation region of an airfoil and influence of wake on film effectiveness where carried out on the BFC rig. Pressure gradient effects are therefore studied on the IFC rig.

**BFC Wind Tunnel Details**

Figure 11 shows the schematic diagram of the experimental apparatus. The apparatus consists of a main flow supply, a secondary flow supply and associated measuring instruments. The 15-kW blower supplies air at a rate of 4.719 m$^3$/s, yielding a velocity of approximately 52 m/s at the test-section inlet. In this closed loop configuration, the work done by the fan heated up the recirculated air to about 66°C (340 K) at the test section inlet over a period of about 3 hrs as it reached steady state. The blower outlet was connected to a flow-conditioning module consisting of a honeycomb, and three screens. The flow-conditioning module followed by a two-
dimensional nozzle is connected to the inlet of the test-section. The test-section was of 1.2 m length, 0.53 m wide and 0.154 m height. Downstream of the test-section, a two-dimensional diffuser with an area ratio of 3.5 and a total length of 2.1 m serves to recover the pressure in the tunnel. The test-section was made up of a 12.7-mm thick transparent Plexiglas plates, while and the test coupon was made of the same material. Free stream turbulence intensity is less than 1% at the test section.

The secondary flow supply was nitrogen gas. Nitrogen gas is obtained from the boil off of liquid nitrogen contained in dewars (large storage, vacuum insulated tanks) commonly at 1.62 MPa. Using a system of valves, and liquid nitrogen’s natural thermal instability, gas flow is obtained at controllable rates. The temperature at which the gas exits the vessel depends on the mass flow rate of the gas; the larger the amount being released, the colder the gas’ temperature.
If a large enough amount of gas is released continuously, the resulting outflow will be liquid. Therefore, while running a test, in order to achieve nitrogen gas flow at a desired temperature, the mass flow rate must be monitored. Nitrogen gas exits the tank through a main-flow control valve and into an insulated 1.27 cm diameter copper pipe. The copper pipe runs an approximate length of 7 meters before reaching a plenum.

Multiple instruments were used to capture the flow conditions of the mainstream and coolant flow. Type E thermocouples were placed in multiple locations of the BFC rig and the plenum in order to monitor and validate the temperature of the mainstream and the coolant while the rig warmed up, and while TSP images were being captured. Figure 12 shows the location of the thermocouples.

![Figure 12: Location of thermocouples in BFC rig](image)

The thermocouple in location 1 was used primarily to measure the mainstream temperature. This temperature started at about 25 °C and could reach up to 69 °C in 2.5 hours. The thermocouple in location 3 was also used to monitor the temperature of the mainstream, but it always registered the same temperature of location 1, so it was seldom used. The thermocouple
in location 2 was used to monitor the recovery temperature of the test section floor. While the mainstream could warm up in as little as two hours to 62ºC, location 2 took longer to catch up to the mainstream temperature. The temperature reading takes approximately three and a half hours to stabilize since the rig heats up very slowly. When steady state is reached in the rig, the difference between the temperature registered on the floor of the test section and that of the mainstream becomes about 1.5 ºC and remains quite steady throughout the test. The uncertainty in measurements with these thermocouples is 1.0ºC.

Pressure measurements were made with pressure taps connected to a Scanivalve pressure transducer. The range of the transducer is from -34.5 kPa to 34.5 kPa, and has a sensitivity of 6.9 Pa (0.001 psi). It is connected to the plenum through a pressure tap located on the side of the plenum, and was used to monitor coolant static pressure. Other measurements involved the static and dynamic pressure of the test section, performed regularly to assure tunnel stability and to characterize the velocity profile at the injection point. Figure 13 shows the NIST-certified calibration of the Scanivalve.

![Scanivalve Calibration Curve](image)

Figure 13: Scanivalve Calibration Curve
Flow measurements were made with two different thermal mass flow meters, high range and low range to cover the entire range of testing, and keep the accuracy as high as possible. The high range meter was a SIERRA 730 Series Accu-Mass thermal flow meter with a range of 0-1100 L/min, a response time of 200 ms, and an accuracy of 1% of full scale. The low range flow meter used was a McMillan 50K-14C, with a range of 0-500 SCFH, also with an accuracy of 1% of full scale. During testing, it was ensured that the company-recommended tubing schemes were followed. Tests were performed first with the low flow rate, for the lower pressure ratios, and then with the high flow meter to cover the high pressure ratios.

The test coupon used in this investigation was machined in acrylic and consisted of 12 cylindrical holes inclined at 35º with a length to diameter (L/D) ratio of 7.5 and pitch to diameter (P/D) ratio equal to 3. The test coupon is depicted in Figure 15.
An axisymmetric airfoil and rectangular plate was machined out of acrylic. During testing, the airfoil was placed downstream of the injection point and its distance was varied to simulate the different film cooling rows preceding the leading edge of the airfoil. As shown in Figure 16, the leading edge radius-to-hole pitch ratio is 5. The airfoil position is varied along the streamwise direction. This will provide data for a variety of closeness of the stagnation region to the injection point.

Hot gases from the combustor pass through the transition pieces entering the first stage vane. The adjoining walls of the transition pieces induce wake into this stage. In this study wake is induced by using a rectangular acrylic plate, which is placed upstream of the injection location. The wake plate is 0.5 inches thick and 5 inches long. Its trailing edge is rounded on
the corners to reflect the adjoining walls of the transition pieces. The wake plate location is also varied to study the effect of the strength of the wake on film cooling effectiveness.

Figure 16: Leading Edge radius to Film Cooling Hole Pitch Ratio

Figure 17: Test Setup of Airfoil with Wake plate

Twelve different configurations were tested, including the baseline cases. Table 1 lists the different configurations studied in this investigation. Schematic of these configurations are also depicted in Figure 18.
Figure 18: Test Configurations for study on BFC

Table 1: Test Matrix for BFC Tunnel

<table>
<thead>
<tr>
<th>Nomenclature</th>
<th>Distance from Airfoil to LE (x/d)</th>
<th>Distance from Wake Plate to LE (x/d)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0-a</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>0-b</td>
<td>-</td>
<td>12.7</td>
</tr>
<tr>
<td>0-c</td>
<td>-</td>
<td>50.8</td>
</tr>
<tr>
<td>1-a</td>
<td>6.4</td>
<td>-</td>
</tr>
<tr>
<td>1I-a</td>
<td>12.7</td>
<td>-</td>
</tr>
<tr>
<td>1II-a</td>
<td>25.4</td>
<td>-</td>
</tr>
<tr>
<td>1III-a</td>
<td>25.4</td>
<td>-</td>
</tr>
<tr>
<td>1-b</td>
<td>6.4</td>
<td>12.7</td>
</tr>
<tr>
<td>1I-b</td>
<td>12.7</td>
<td>12.7</td>
</tr>
<tr>
<td>1II-b</td>
<td>25.4</td>
<td>12.7</td>
</tr>
<tr>
<td>1III-b</td>
<td>25.4</td>
<td>12.7</td>
</tr>
<tr>
<td>1-c</td>
<td>6.4</td>
<td>50.8</td>
</tr>
<tr>
<td>1I-c</td>
<td>12.7</td>
<td>50.8</td>
</tr>
<tr>
<td>1II-c</td>
<td>25.4</td>
<td>50.8</td>
</tr>
</tbody>
</table>
IFC Wind Tunnel Details

The IFC rig can be mainly divided into two sections, vertical section and the horizontal section. The vertical section provided the secondary flow where as the horizontal section was used as the crossflow or mainflow in this case. The fluid used in both the secondary flow and mainflow was air. The secondary flow section is an open-loop duct where the air was supplied by a compressor, and the mass flow rate entering the tunnel was controlled using a regulator. The mainflow was supplied by a suction blower which has a rating of 645 CFM.

Figure 19: Vertical section of the IFC tunnel
The air supply for the impingement vertical duct is supplied by a compressor which is connected to a tank in parallel that can hold up to 200 psi. The air from the compressor is passed through a dehumidifier to remove the humidity of the air. This dehumidified air is stored in the tank. The pipe line from the tank runs through an oil filter which is equipped with a carbon-activated oil vapor separation filter to filter out the oil vapor from the air. This is necessary because if the oil from the compressor enters the test section, it can cause a deposit on the temperature sensitive paint (TSP) surface and can disturb the paint surface and have a direct impact on the readings. A regulator is installed at the beginning of the piping for the tunnel right after the oil filter. This enables us to set the required mass flow rate through the impingement tunnel. The inlet mass flow rate at the regulator is measured using a Scanivalve inline venturi meter. A pressure tap and a thermocouple are installed at the Scanivalve to calculate the mass flow rate at those particular conditions.
The air after being set at the required mass flow rate conditions is then passed through an 18 KW Sylvania inline air heater which has the highest temperature rating of 1400°F. The heater is installed on the ground and can operate continuously for 5000 hrs or more. A thermocouple is installed one inch after the exit of the heater, the other end of which is connected to the electric heater feedthrough system. The two important factors for the heater are that the element wire should not go above 1066°C for maximum heater life and there must be sufficient flow through the heater so that the epoxy seal at the feed thorough does not become overheated. Since the heater is controlled using a temperature controller, the response time of the thermocouple and temperature controller become critical. The thermocouple should be an exposed junction style, maximum wire diameter of 0.020”, and should be located within an inch of the heater exit and should be located at/or slightly beyond the center of the pipe. The limiting factor for flow is the ability to sense the air temperature. At low flow rates, the thermocouple response will lag far behind the element temperature and there is a potential of overheating the element wire. The overshoot in the element temperature will be more critical if the heater is being run near its maximum temperature than if the heater is being run at relatively low temperatures. As an estimate for thermocouple sensing, a minimum flow of 200 SCFH at lower temperatures and 400 SCFH at higher temperatures would work.

The entrance to the tunnel is of rectangular cross-section with a distributor, which is a rectangular plate with holes so it can distribute the flow from the pipe throughout the rectangular crosssection. The flow is then made to pass through a diffuser for pressure recovery. Honeycombs are set after the distributor for flow distribution and flow straightening. Then the hot air is made to pass through a 3:1 contraction nozzle which is then connected to the plenum located above the
test section. The entire flow conditioning section is insulated and sealed at all the connections to make sure there is no heat or mass loss through it.

Crossflow duct is the section where the target plate with film cooling holes is installed on. The duct as seen in Figure 21 is installed with foam inserts on either side, to create the streamwise pressure gradient. The bottom side of the duct is made of acrylic, so the TSP on the other side of the target plate can be seen for the camera which is fixed underneath the duct.

Only one test configuration was studied. The test plate consisted 5 staggered rows of holes inclined at 30º with a length to diameter (L/D) of 25 and pitch to diameter (P/D) of 8. The test plate was made of Rohacell. Rohacell is polymethacrylimide (PMI-) rigid foam that is used as a core material for sandwich constructions. It shows outstanding mechanical and thermal properties. In comparison to all other foams, it offers the best ratio of weight and mechanical properties as well as highest heat resistance. The WF 200 product is the hardest Rohacell available in the market at the present instance. The main reason for the use of Rohacell in this study is its thermal conductivity value which is as low as 3.3x10-02 Watts/m C.
The inserts which guide the crossflow are made of light weight sturdy foam. The main function of the inserts is to vary the cross-section, which will in turn vary the Mach number along the film cooling hole arrays. The design of the inserts is based on the Mach number variation required. Figure 23 shows the alignment of the film holes with the inserts. The inserts are bolted to the crossflow duct underneath the target plate. Kapton tape is used along the surface of the inserts for smoothness of the flow. Black paint is applied on the Kapton tape, so it does not reflect the light on the paint, while taking the actual test pictures.
Uni-coat Temperature Sensitive Paint, TSP, formulated by ISSI, is used in this study. The effective temperature range is 0-100 °C, beyond which the temperature sensitivity of TSP becomes weaker. It is packaged in aerosol cans and can be applied easily with a spray. After it is heat treated above 100 °C for 30 minutes the temperature sensitivity of the paint is about 0.93ºC, [28]. The TSP painted surface is smooth. The emission spectrum of TSP is shown in Figure 24. An optical 590-nm long pass filter is also used on the camera to separate the excitation light and emission light from the paint.
Figure 24: Emission spectrum of TSP (Liu, 2006)

TSP incorporates luminescent molecules in paint together with a transparent polymer binder. Light of the proper wavelength is directed at the painted model to excite the luminescent molecules. The sensor molecules become excited electronically to an elevated energy state. The molecules undergo transition back to the ground state by several mechanisms, predominantly radiative decay (luminescence). Sensor molecules emit luminescent light of a longer wavelength than that of the excitation light. The appropriate filters can separate excitation light and luminescent emission light, and the intensity of the luminescent light can be determined using a photodetector. The excited energy state can also be deactivated by quenching processes. Through two important photo-physical processes known as thermal- and oxygen-quenching, the luminescent intensity of the paint emission is inversely proportional to local temperature. In principle, a full spatial distribution of the surface temperature can be obtained by using the TSP technique.
A high resolution 14-bit CCD (Charged Couple Device) camera was utilized for this study. It is a PCO-1600 CCD camera, depicted in Figure 26, provided by the Cooke Corporation with spatial resolution of 1200 by 1600 pixels. The image data is transferred via an IEEE 1394 (“firewire”) cable and firewire PCI card to a data collection PC. “CamWare” software provided by Cooke Corp. is used in the Windows operating system to control initialization, exposure time and image acquisition. The acquired image data are processed using MATLAB. The camera is thermo-electrically cooled and has a high quantum efficiency at the paint emission wavelengths. The choice and quality of the scientific-grade camera dictate the measurement accuracy.

LED-based illumination source (peak wavelength at 464 nm) was selected as the excitation light for the TSP. The stability of the light source provided by ISSI is within 1% after 10 minute warm up. The CCD camera and excitation spectrum of LED is shown in Figure 26.
Figure 26: CCD camera and light source

Figure 27: Spectrum of LED source (Liu, 2006)
Computational Methodology

Computational Fluid Dynamics (CFD) plays an important role in the design of engine components. In this study a comprehensive methodology was established and implemented to consistently and more accurately predict film-cooling behavior. The common traits shared by all computations documented in this work are described in detail in this section that follow; the simulation specifics are documented in the appropriate sections.

Computational Model

In any computational simulation the accuracy of the results will depend strongly on the application of a model to the problem at hand. The computational models match the experimental test cases. Special attention is paid to the film hole region; due to the highly complex recalcitrating nature of this area.

Due to the height of the wind tunnel used in these experiments, a symmetry condition was applied at the top of the domain. A pressure profile was applied and it was adjusted accordingly to match the boundary layer thickness at the location of the inlet plane. The velocity was measured from experimental results and then it was used to estimate the inlet pressure profile shown in Figure 29. A FORTRAN subroutine was then written to read this profile into FLUENT and impose it as a boundary condition. An outlet pressure boundary condition was applied far downstream from the jet injection site. At the plenum inlet, mass flow inlet conditions were imposed.
Pressure Inlet
- from measured Velocity and Turbulence

Mass Flow Rate
- Nitrogen modeled as “Air”

Pressure Outlet
- Static Pressure at the exit

Figure 28: Computational Domain and Boundary Conditions Type (Configuration 0-a)

Figure 29: Inlet Total Pressure Profile
Grid and Mesh Generation

A multi-block numerical grid was used in all cases to create a structured mesh. The grid was created in Gambit from Fluent, Inc. The cells in the near-wall layers were stretched away from the surfaces, and the first mesh point above the endwall was chosen such that the average is of the order of unity. High cell concentration was placed where large flow/or temperature gradients are expected.

After the grid generation phase is complete, the mesh is imported into FLUENT. The flow is initialized according to the boundary conditions obtained from the experimental conditions. Grid independence is established by increasing the number of cells in the grid and performing test simulations until a negligible change in all field and surface results is accomplished. For some cases, the maximum grid density was limited by the capabilities of the computational resources.

Solution and Discretization

The solution of the computational domain is performed using FLUENT software package from Fluent, Inc. The use of second order schemes were used in this study to provide the highest accuracy available. Multigrid was used to decrease the CPU time required for each simulation. The multigrid procedure solves the domain on varying levels of grid density smaller than the original grid, thereby allowing information to propagate over the domain more quickly.
Turbulence Model

The most obvious method to resolve turbulent flow effects is to solve the Navier-Stokes equations directly. However, the amount of computational resources necessary to handle the degree of grid resolution is prohibitive. A practical method must be employed to model the effects of turbulence both in the simulation performed in the work and in a potential design tool that gas turbine designers can use.

This study employed the Realizable \( \kappa-\varepsilon \) model with enhanced wall treatment for all its cases. It differs from the standard \( \kappa-\varepsilon \) model in two important ways: (1) contains a new formulation for the turbulent viscosity. (2) A new transport equation for the dissipation rate, \( \varepsilon \), has been derived from an exact equation for the transport of the mean-square vorticity fluctuation. The term "realizable" means that the model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows. Analysis have shown that Realizable \( \kappa-\varepsilon \) is a better predictor of spreading rate of both planar and round jets. It is also likely to provide superior performance for rows involving rotation, boundary layers under strong adverse pressure gradients, separation, and recirculation.

The modeled transport equations for \( \kappa-\varepsilon \) in the realizable \( \kappa-\varepsilon \) model are:

\[
\frac{\partial}{\partial t} \rho \kappa + \frac{\partial}{\partial x_j} (\rho \kappa u_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_k}{\sigma_\kappa} \right) \frac{\partial \kappa}{\partial x_j} \right] + G_\kappa + G_b - \rho \varepsilon - Y_\mu + S_\kappa \tag{Equation 6}
\]

\[
\frac{\partial}{\partial t} \rho \varepsilon + \frac{\partial}{\partial x_j} (\rho \varepsilon u_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_k}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 \varepsilon S_k - \rho C_2 \frac{\varepsilon^2}{\kappa + \sqrt{\nu \varepsilon}} + C_4 \frac{\kappa}{\varepsilon} - C_{\kappa} G_\kappa + S_\varepsilon \tag{Equation 7}
\]
where,

\[
C_1 = \max\left(0.43, \frac{\eta}{\eta + 5}\right) \quad \text{(Equation 8)}
\]

\[
\eta = S \frac{\kappa}{\varepsilon} \quad \text{(Equation 9)}
\]

\[
S = \sqrt{2S_\alpha S_\gamma} \quad \text{(Equation 10)}
\]

The coefficients were set as default by the software.

*Near-Wall Treatments*

Turbulent flows are significantly affected by the presence of walls. Obviously, the mean velocity field is affected through the no-slip condition that has to be satisfied at the wall. However, the turbulence is also changed by the presence of the wall in non-trivial ways. Very close to the wall, viscous damping reduces the tangential velocity fluctuations, while kinematic blocking reduces the normal fluctuations. Toward the outer part of the near wall region, however, the turbulence is rapidly augmented by the production of turbulence kinetic energy due to the large gradients in mean velocity.

The near-wall modeling significantly impacts the fidelity of numerical solutions, in as much as walls are the main source of mean vorticity and turbulence. Therefore, accurate representation of the flow in the near-wall region determines successful predictions of wall-bounded turbulent flows.

Enhanced wall treatment is a near-wall modeling method that combines a two-layer model with enhanced wall functions. If the near-wall mesh is fine enough to be able to resolve
the laminar sublayer (typically \( y^+ \approx 1 \)), then the enhanced wall treatment will be identical to the traditional two-layer zonal model (see below for details). However, the restriction that the near-wall mesh must be sufficiently fine everywhere might impose too large a computational requirement. Ideally, then, one would like to have a near-wall formulation that can be used with coarse meshes (usually referred to as wall-function meshes) as well as fine meshes (low-Reynolds-number meshes). In addition, excessive error should not be incurred for intermediate meshes that are too fine for the near-wall cell centroid to lie in the fully turbulent region, but also too coarse to properly resolve the sublayer.

To have a method that can extend its applicability throughout the near-wall region (i.e., laminar sublayer, buffer region, and fully-turbulent outer region) it is necessary to formulate the law-of-the-wall as a single wall law for the entire wall region. This is achieved by blending linear (laminar) and logarithmic (turbulent) laws-of-the-wall. This approach allows the fully turbulent law to be easily modified and extended to take into account other effects such as pressure gradients or variable properties. The first layer of cells must have a cell center with \( y^+ \) measured by the \( y^+ \), where \( y^+ \) is defined as:

\[
y^+ = \frac{\rho u_r y}{\mu}
\]

(Equation 11)

Building a grid to meet \( y^+ \) requirements leads to some serious implications in reaching grid independence and maintaining high quality grid. Near-wall approach is adequate for these cases were the low Reynolds number effect is pervasive in the flow domain. All cases required a near wall treatment that is valid in the viscosity-affected region and accordingly integrable all the way to the wall.
Grid Independence

A grid independence check was carried out to determine the proper mesh size. The test was carried out on the baseline case for that consisted of one cylindrical holes. The centerline temperature was evaluated to determine if independence was achieved. Grid independence is achieved when temperature variations from one grid size to another was minimal. For some cases, the maximum grid density was limited by the capabilities of the computational resources.

![Grid Independence Check](image)

Convergence

Convergence was determined with the following strict criteria: (1) Reduction of all residuals of at least 4 orders of magnitude (2) Global mass and energy imbalance below 0.01% and (3) Flow field and endwall surface temperature invariant with additional iterations.
Data Reduction of Experimental Data

Discharge Coefficient

In a orifice or other constriction, the ratio of the mass flow rate at the discharge end of the orifice to that of an ideal orifice which expands an identical working fluid from the same initial conditions to the same exit pressure is also known as discharge coefficient. In other words, the discharge coefficient compares the amount of flow passing through an orifice to the theoretical flow, based on the compressible flow equations for a specific area. The discharge coefficient depends on the geometry of the constriction and flow conditions.

The amount of mass flow through the coupon at increasing pressure ratios was measured. The tunnel has warmed up, the pressure in the plenum is increased at very small intervals and the mass flow rate is taken for each pressure. For this test, the secondary flow supply was air. Coolant temperature, static pressure and mainstream temperature are the main data taken for this test. The tests were performed without the airfoil, wake plate or side inserts. Once the highest pressure has been recorded, the tunnel can be stopped. Pressure and mass flow rate biases are averaged between the beginning of the test and the end.

In the BFC rig, the discharge coefficient can be directly calculated using the following equation:

\[
C_D = \frac{\frac{m}{N}}{\frac{\pi}{4} \cdot D^2 \cdot P_e \left( \frac{P_{stat}}{P_e} \right)^\frac{k-1}{k} \sqrt{\frac{2 \cdot k \cdot \frac{1}{R \cdot T_e}}{k-1} \left[ \frac{P_e}{P_{stat}} \right]^\frac{k-1}{k}}}
\]

(Equation 12)
During these tests, the measured variables were: the volumetric flow rate, which was then multiplied by the density to yield the mass flow rate, \( m_a \); \( T_c \), the temperature of the coolant inside the plenum; \( P_c \) the static pressure of the coolant inside the plenum; and \( P_{\text{stat}} \), the main flow static pressure, at the test section. Since the crosssection of the test section is rectangular, no additional pressure measurements of the mainstream were necessary. The remaining terms, \( N \), \( \kappa \), \( R \), and \( D \) were know constants, or obtained from tables.

On the other hand, in the IFC tunnel the pressure distribution along the inserts was calculated by using the Mach number values, based on the area at the crosssection of each hole exit rows. Pressure distribution is then obtained by:

\[
\frac{P}{P_t} = \left(1 + \frac{\gamma - 1}{2} \cdot M^2 \right)^{-\frac{\gamma}{\gamma - 1}}
\]

(Equation 13)

where \( P \) is the pressure, \( t \) indicates total conditions, \( \gamma \) is the specific heat ratio and \( M \) is the Mach number.

Figure 31: Mach number distribution along the film hole rows
Using this pressure distribution along the inserts and using Bernoulli’s equations, the average velocity at the exit of each row of holes were calculated.

\[ P + \frac{1}{2} \cdot \rho \cdot V^2 + \rho \cdot g \cdot h = \text{const} \]  
(Equation 14)

Neglecting \( h \) and solving for velocity gives,

\[ V = \sqrt{\frac{2 \cdot (P_2 - P_1)}{\rho}} \]  
(Equation 15)

The calculated velocity was then used to calculate the theoretical mass flow rate by:

\[ m = \rho \cdot A_h \cdot V \cdot N_h \]  
(Equation 16)

where \( A_h \) and \( N_h \) indicates the crossectional area of the hole and number of holes.

Actual mass flow rate is obtained from the mass flow rate reading and CD is calculated by the ratio of the actual mass flow rate to the theoretical value.

\[ C_D = \frac{\dot{m}}{m_{th}} \]  
(Equation 17)

**Blowing Ratio and Momentum Flux**

The blowing ratio, also called the mass flux ratio, is a dimensionless number used in film cooling to quantify the ratio of the mass flow rate per unit area of the coolant to that of the mainstream.
Given that,

\[ m_c = \rho_c \cdot A_h \cdot V_c \cdot N_h \quad \text{(Equation 19)} \]

The blowing ratio can also be expressed as,

\[ \frac{m_c}{\rho} \cdot \frac{A_c}{A_m} = \frac{\rho_c \cdot V_c}{\rho_m \cdot V_m} \quad \text{(Equation 18)} \]

Another parameter that is often used is the momentum flux ratio. The momentum flux, I, compares the kinetic energy of the coolant (per volume) versus that of the mainstream. The momentum flux ratio is defined as:

\[ I = \frac{\rho_c \cdot (V^2)_c}{\rho_m \cdot (V^2)_m} \quad \text{(Equation 21)} \]

Since the density ratio, DR, is kept nearly the momentum flux ratio can also be obtained by:

\[ I = \frac{M^2}{DR} \quad \text{(Equation 22)} \]

**Film Cooling Effectiveness**

Film cooling effectiveness (\( \eta \)) measurement is the main parameter used for comparison in this study. As mentioned in Chapter 1 effectiveness is defined as:
\[ \eta = \frac{T_r - T_{aw}}{T_r - T_c} \]  

(Equation 23)

However, obtaining the final plots of \( \eta \) requires a significant amount of data reduction. During the test, three parameters are recorded: \( T_c \) – the coolant temperature, \( T_{aw} \) – the adiabatic wall temperature distribution downstream of the cooling holes, and \( T_r \) – the recovery temperature of the test section. To analyze the TSP pictures, matrix handling software such as MATLAB is used. The images are read into a matrix containing the intensity information over every pixel of the TSP.

Figure 32 shows an image of the TSP downstream of the coolant holes. This image was taken under non-test conditions, without any LED lighting, and is used mainly for locating the holes with respect to the edges of the paint layer. The coolant holes are very difficult to see under testing conditions because they reflect back very little light. Figure 33 shows the same image, but this time under LED lighting. If the TSP is at room conditions, and testing is ready to begin, then this image becomes the reference image, and the intensity from any pixel becomes \( I(TR) \).
LED light is kept on throughout the duration of the test. Neither the camera nor the LED light may be moved, or the usage of the reference image will not be valid due to changes in lighting conditions. Precautions must be taken to ensure that the data will not be polluted due to outside lighting condition changes. For all tests and blowing ratios, images are taken in sets of four, which are then averaged to filter out any fluctuations. For the reference image, the temperature must be uniform, and known. Typical reference temperatures are around 25 °C, or room temperature.

![Reference Image](image)

Figure 33: Reference Image

The tunnel is allowed to warm up for over four hours. At that point another set of pictures is taken, named BR0, referring to zero blowing ratio. To obtain a flow image, first the plenum must be kept at a constant pressure and temperature for approximately 8-10 minutes to achieve the desired blowing ratio, and thus film cooling conditions over the paint. Once these conditions are met, the set of images can be taken. Figure 34 shows TSP under cooling conditions.
The process of taking images continues until all the blowing ratios are taken. Once this has occurred, the testing ends and the images can be processed. Temperature is obtained as follows: the ratio of emission intensity I(T) at any temperature T to the emission intensity I(TR) at an unspecified reference temperature TR is,

$$IR = \frac{I(T)}{I(TR)} = fn(T, TR)$$  \hspace{1cm} (Equation 24)

Local ratios are then converted into temperature values. This is accomplished with the use of the calibration curve of the paint. Figure 35 shows a typical calibration curve for TSP.
The manipulation is done with software on a per-pixel level. Once the processing is done, the intensity information yields temperatures. This is how the recovery temperature and the adiabatic wall temperature distributions are obtained.

![Figure 36: Raw Temperature image from TSP](image)

The area captured by the CCD camera is significant in size, and does not provide the resolution to calculate centerline effectiveness. Instead average effectiveness is calculated. To calculate the span-wise-averaged cooling effectiveness, several physical factors are considered such as edge effects, the number of cooling holes and the diameter of the holes. To discount edge effects from influencing the temperature data, only holes close to the middle of the coupon center are included in the averaging step.

The intensity data is reduced to pixel temperature data and local film cooling effectiveness is calculated over the TSP surface. The data is then averaged in the spanwise direction yielding an effectiveness distribution along the streamwise direction. The result of this process is shown in Figure 37.
Figure 37: Results of averaging effectiveness

Uncertainty Analysis

The uncertainties reported were estimated following the procedure described by Kline and McClintock (1953). The effectiveness as defined in the equation below is used to find the derivatives in the uncertainty equation:

\[
U_n = \sqrt{\left(\frac{d}{dT_e} \eta \cdot U_{\eta e}\right)^2 + \left(\frac{d}{dT_r} \eta \cdot U_{\eta r}\right)^2 + \left(\frac{d}{dT_{aw}} \cdot \eta \cdot U_{\eta r}\right)^2}
\]  

(Equation 25)

In this case UT_r and UT_{aw}, the uncertainties in the recovery and adiabatic wall temperatures are equal since both are obtained with TSP, which was calibrated with a thermocouple. U_{Tc} is 1.0°C, which is the uncertainty in the coolant temperature, as measured with the plenum thermocouple set up. The values of U_{\eta} are evaluated at every point on the effectiveness curve, and give the uncertainty band shown in Figure 38.
The uncertainty in the discharge coefficient has been calculated similarly:

\[
U_{CD} = \sqrt{\left(\frac{d}{dm} C_D \cdot U_m\right)^2 + \left(\frac{d}{dP_{stat}} C_D \cdot U_p\right)^2 + \left(\frac{d}{dP_e} C_D \cdot U_p\right)^2 + \left(\frac{d}{dT_{avg}} C_D \cdot U_t\right)^2 + \left(\frac{d}{dD_{avg}} C_D \cdot U_D\right)^2}
\]

(Equation 26)

This equation yields results that suggest the uncertainty at the lower pressure ratios is larger than the uncertainty at the higher pressure ratios.
Considering both rigs, it can be summarized that the uncertainties of the measurements in the present study are as follow: (1) Precision interval of TSP is found to be ±0.93°C with 95% level of confidence [28] (2) The uncertainty in the effectiveness values was on an average less than 3%, while the highest value was calculated to be less than 9%. (3) Uncertainty of CD values ranges in the order of ± 9%, and ± 1% at the higher pressure ratios.

**Data Reduction of Computational Data**

*Film Cooling Effectiveness*

The average effectiveness was calculated using FLUENT predictions by exporting the temperature data from the bottom adiabatic wall of the model. The data is the processed using a code written in C++. The execution of the code consists of arranging the raw data by x, z and corresponding temperature value. It then averages temperature and calculates average effectiveness. A GUI (Graphical User Interface) was created to facilitate the process of the simulation data. The process flowchart explains the details of the GUI and execution of the code.
Figure 40: FLUENT Local Temperature Data extracted for calculating effectiveness
Figure 41: Graphical User Interface to process Computational Data
Figure 42: Process flowchart of Data Reduction of Computational Data

**GUI EXECUTION DETAILS**

- Create input file by Outputting temperature data from FLUENT
- Initialize parameters (D, Tp, Tr, digits of precision of input and output file)

**CODE EXECUTION DETAILS**

- Read input file and remove all negative x’s (x = 0 @ trailing edge of film holes)
- Remove columns not needed (cellnumber, y-coordinate) and reformat file to maintain digits of precision
- List x’s and z’s unique values and corresponding Temperature values.
- Average temperature and calculate η

\[ \eta = \frac{T_r - T_{\text{mean}}}{T_r - T_{\text{wall}}} \]

**Input file**

<table>
<thead>
<tr>
<th>Temperature</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.6000</td>
</tr>
<tr>
<td>0.6250</td>
</tr>
<tr>
<td>0.6500</td>
</tr>
<tr>
<td>0.6750</td>
</tr>
<tr>
<td>0.7000</td>
</tr>
<tr>
<td>0.7250</td>
</tr>
<tr>
<td>0.7500</td>
</tr>
<tr>
<td>0.7750</td>
</tr>
</tbody>
</table>

**Formatted file**

<table>
<thead>
<tr>
<th>Coordinate</th>
<th>0.0000</th>
<th>0.0020</th>
</tr>
</thead>
<tbody>
<tr>
<td>Temperature</td>
<td>0.8000</td>
<td>0.8125</td>
</tr>
<tr>
<td></td>
<td>0.8250</td>
<td>0.8375</td>
</tr>
<tr>
<td></td>
<td>0.8500</td>
<td>0.8625</td>
</tr>
</tbody>
</table>

**Sorted data**

<table>
<thead>
<tr>
<th>Coordinate</th>
<th>0.0000</th>
<th>0.0020</th>
</tr>
</thead>
<tbody>
<tr>
<td>Temperature</td>
<td>0.8000</td>
<td>0.8125</td>
</tr>
<tr>
<td></td>
<td>0.8250</td>
<td>0.8375</td>
</tr>
<tr>
<td></td>
<td>0.8500</td>
<td>0.8625</td>
</tr>
</tbody>
</table>

**Output data**

<table>
<thead>
<tr>
<th>Coordinate</th>
<th>0.0000</th>
<th>0.0020</th>
</tr>
</thead>
<tbody>
<tr>
<td>Temperature</td>
<td>0.8000</td>
<td>0.8125</td>
</tr>
<tr>
<td></td>
<td>0.8250</td>
<td>0.8375</td>
</tr>
<tr>
<td></td>
<td>0.8500</td>
<td>0.8625</td>
</tr>
</tbody>
</table>

**NO**

- Is the Airfoil present?
  - NO
  - Check “Create Bounded Eta” and select corresponding streamline case
  - Browse for input file “Set input file”
  - Click “Run” (execute code)

**YES**

- Check “Create Bounded Eta” and select corresponding streamline case
- Browse for input file “Set input file”
- Click “Run” (execute code)
CHAPTER 4: RESULTS

Baseline

The baseline case consists the test coupon of a single row of cylindrical holes inclined at 35° with a L/D = 7.5 and a PI/D = 3. The test was conducted as described in Chapter 3. This section describes the experimental readings and computational details specific to this configuration.

Experimental Details

During these tests, the measured variables were the $T_c$, the temperature of the coolant inside the plenum; $P_c$ the static pressure of the coolant inside the plenum to match target mass flow rate based on discharge coefficient test; and $P_{stat}$, the main flow static pressure, at the test section. Tests were carried out for four blowing ratios, $M=0.5, 0.75, 1.0, 1.5$. The test conditions are described in Table 2.

CFD Details

The numerical model consisted of 726,450 cells. The cell topology is hexahedral and $y^+$ was close to unity. As illustrated in Figure 44 the side and top walls were defined as symmetry plane where as a pressure boundary condition was prescribed at the inlet and outlet of mainstream. On the plenum, a mass flow inlet was specified. The numerical simulation was
carried out for the lowest and highest blowing ratios (M=0.5 and 1.5). The values used as boundary conditions matched the experimental measurements.

Table 2: Baseline Test Experimental Measurements

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Mainstream Conditions</th>
<th>Plenum Conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>M = 0.5</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>337</td>
<td>260</td>
</tr>
<tr>
<td>Mass flow rate (average per hole)</td>
<td>-</td>
<td>8.49 x 10^{-5}</td>
</tr>
<tr>
<td><strong>M = 0.75</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>337</td>
<td>255</td>
</tr>
<tr>
<td>Mass flow rate (average per hole)</td>
<td>-</td>
<td>1.22 x 10^{-4}</td>
</tr>
<tr>
<td><strong>M = 1.0</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>337</td>
<td>255</td>
</tr>
<tr>
<td>Mass flow rate (average per hole)</td>
<td>-</td>
<td>1.63 x 10^{-4}</td>
</tr>
<tr>
<td><strong>M = 1.5</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>337</td>
<td>256</td>
</tr>
<tr>
<td>Mass flow rate (average per hole)</td>
<td>-</td>
<td>2.45 x 10^{-4}</td>
</tr>
</tbody>
</table>
Figure 43: Configuration 0-a CFD Grid

**Pressure Inlet**
- from measured Velocity and Turbulence

**Mass Flow Rate**
- Nitrogen modeled as “Air”

**Pressure Outlet**
- Static Pressure at the exit

Figure 44: CFD Domain and Boundary Conditions Type for Configuration 0-a
Results and Discussion

Average film cooling effectiveness for the baseline configuration was compared against existing literature. The present data is compared with the data of Sinha et al. [8] and Lutum [10] for two different blowing ratios. Figure 45 and Figure 46 show that the presented experimental results correlate well with the results presented in the literature validating the procedure followed to acquire the data for the present study.

Figure 45: Data Comparison for M=0.5
The experimental data was also compared to the numerical predictions. Researchers have used different turbulence models to predict average film cooling effectiveness yielding different results. As discussed in Chapter 2, it is very common to use RKE or SKW model for modeling film cooling. At the beginning of this study it was unknown which of the two models could be use for the analysis that could better predict average effectiveness, therefore, both models were employed and the results showed that based on our experimental data RKE better predicts film cooling effectiveness. Figure 47 shows the comparison of the two turbulence models and experimental data for $M = 0.5$. Film cooling effectiveness predictions for RSW model are valid for $x/d \leq 10$ where as for RKE model the predictions are valid up to $x/d \leq 25$. Using the RKE model, for $x/d=50$ the difference between the numerical predictions and experimental measurements could be as high as 27%. The error calculated using the SKW model is estimate.
to be as high as 30% for the same x/d. The numerical predictions of film cooling effectiveness were also compared against Lutum [10] and Sinha [8] experimental data. The comparison shows that RKE is also a better predictor of film cooling effectiveness. Literature has shown that SKW may over predict the diffusion of jets, which explains the high effectiveness value at higher x/d. For this reason, the numerical simulation used RKE model as a turbulence model for predicting film cooling effectiveness.

Figure 47: Comparison of Turbulence Models against Experimental Data
Figure 48: Validation of Configuration 0-a

Figure 49 through Figure 51 shows average effectiveness for varying blowing ratio and local effectiveness distribution for Configuration 0-a. The results show that for low blowing ratio higher effectiveness occurs close to the injection point. For high blowing ratio effectiveness decreases due to jet lift off. Jet to jet interaction is promoted as a result of an increase of the blowing ratio. The interaction of the jet with the nearby jets and mainstream enhances mixing resulting in a slight increase in effectiveness.
Figure 49: Average Effectiveness for Configuration 0-a

Figure 50: Local Temperature Distribution for Configuration 0-a

Figure 51: Local Temperature Distribution for Configuration 0-a (Exp vs. CFD)
The computational results agree with the experimental data. Streamline case shows that for low blowing ratios, the jet remains close to the surface. The opposite is observed with the higher blowing ratio case. Figure 53 shows the detachment of the flow as it enters the mainstream. The flow later reattaches resulting in an increase in the average film effectiveness.

Sinha et al. [8] observed that the laterally averaged effectiveness is strongly dependent on the lateral spreading of the cooling jet which increases laterally average effectiveness. Similarly, Eriksen and Goldstein [6] observed that higher blowing ratios promote the interaction of the jets with the adjacent jets and with the mainstream flow resulting in a higher effectiveness farther downstream of the injection point. These findings are consistent with the results of the present study. Higher effectiveness promotes jet interaction resulting in higher effectiveness when reattachment of the jet occurs.

Figure 52: Streamline Plot for Configuration 0-a (M=0.5)
A study on film cooling effectiveness was then carried out to understand the effect of the closeness of the stagnation region of an airfoil. The configurations considered in this section are configurations I-a, II-a and II-a. An axisymmetric airfoil is placed downstream of the injection point and temperature distribution was obtained for various blowing ratios. This section describes the experimental readings and computational details specific to these configurations.
Experimental Details

During these tests, the measured variables were the $T_c$, the temperature of the coolant inside the plenum; $P_c$ the static pressure of the coolant inside the plenum; and $P_{stat}$, the main flow static pressure, at the test section. Tests were carried out for four blowing ratios, $M=0.5, 0.75, 1.0, 1.5$. The test conditions are listed in Tables 4-6.
Table 4: Experimental Measurements for Configuration I-a

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Mainstream Conditions</th>
<th>Plenum Conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>$M = 0.5$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$P_{\text{static}}$ (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>337</td>
<td>258</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>101338</td>
</tr>
<tr>
<td>$M = 0.75$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$P_{\text{static}}$ (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>337</td>
<td>255</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>102069</td>
</tr>
<tr>
<td>$M = 1.0$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$P_{\text{static}}$ (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>337</td>
<td>258</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>103241</td>
</tr>
<tr>
<td>$M = 1.5$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$P_{\text{static}}$ (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>337</td>
<td>257</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>105448</td>
</tr>
</tbody>
</table>
Table 5: Experimental Measurements for Configuration II-a

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Mainstream Conditions</th>
<th>Plenum Conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>M = 0.5</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$P_{\text{static}}$ (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>338</td>
<td>255</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>101345</td>
</tr>
<tr>
<td><strong>M = 0.75</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$P_{\text{static}}$ (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>338</td>
<td>261</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>102090</td>
</tr>
<tr>
<td><strong>M = 1.0</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$P_{\text{static}}$ (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>338</td>
<td>255</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>103234</td>
</tr>
<tr>
<td><strong>M = 1.5</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$P_{\text{static}}$ (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>338</td>
<td>257</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>105441</td>
</tr>
</tbody>
</table>
Table 6: Experimental Measurements for Configuration III-a

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Mainstream Conditions</th>
<th>Plenum Conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>M = 0.5</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>339</td>
<td>258</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>101345</td>
</tr>
<tr>
<td><strong>M = 0.75</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>339</td>
<td>260.9</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>102076</td>
</tr>
<tr>
<td><strong>M = 1.0</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>339</td>
<td>261</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>103247</td>
</tr>
<tr>
<td><strong>M = 1.5</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>339</td>
<td>261</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>105523</td>
</tr>
</tbody>
</table>
Simulation Details

Two different cases were modeled in CFD: (1) Configuration 1-a and (2) III-a. The grid size ranged from 1,160,915 to 1,218,098 hexahedral cells. The side and top walls were defined as symmetry plane where as a pressure boundary condition was prescribed at the inlet and outlet of mainstream. On the plenum, a mass flow inlet was calculated based on the pressure reading from the experimental runs. The numerical simulation was carried out for the lowest and highest blowing ratios (M=0.5 and 1.5). The values used in the boundary conditions match the experimental measurements listed in the previous section.

Figure 55: Stagnation Region Effects CFD Grid
Results and Discussion

The stagnation gradient created by the blockage presented by the leading edge of the airfoil causes the flow to undergo three-dimensional separation. A detailed study on film cooling effectiveness was then carried out to understand the effect of the stagnation gradient and closeness of the stagnation region of an airfoil. Figure 57 through Figure 62 show effectiveness measurements obtained for configuration I-a, II-a, III-a. The data shows that for low blowing ratios effectiveness is higher specifically in the region close to the injection point. On the contrary, higher effectiveness is observed at larger x/D.

At lower blowing ratios, jet spreading is minimal, making it necessary to implement alternative means of endwall cooling in the region adjacent to the airfoil. The jet to jet interaction is small and hence there is no spreading of the jet. This jet spreading is more evident
as the blowing ratio is increased. The local effectiveness results shown on Figures 58, 60, and 62 indicate that the lateral spreading of the jets as the blowing ratio is increased.

The stagnation gradient created by the blockage presented by the leading edge of the airfoil causes the flow to undergo three-dimensional separation. The effect caused by the stagnation region not only impacts effectiveness but film coverage. The results also reveal that the area coverage significantly varies with the blowing ratio. This trend was observed for all configurations.

![Figure 57: Average Film Cooling Effectiveness for Configuration I-a](image)

![Figure 58: Local Film Cooling Effectiveness for Configuration I-a (Exp. vs. CFD)](image)
Figure 59: Average Film Cooling Effectiveness for Configuration I-b

Figure 60: Local Film Cooling Effectiveness for Configuration I-b (Experimental)
Figure 61: Average Film Cooling Effectiveness for Configuration I-c

Figure 62: Local Film Cooling Effectiveness for Configuration I-c (Exp. vs. CFD)
Figure 63: Average Film Cooling Effectiveness (Configuration 0, I, and III-a)

Figure 64: Streamline Plot for Configuration I-a (M=0.5 and 1.5)

Figure 65: Streamline Plot for Configuration I-c (M=0.5 and 1.5)
From the streamline plots depicted in Figures 64 and 65 show that for $M = 0.5$ the film remains close to the surface and flows around the airfoil. As the blowing ratio is increased, the jet overcomes the mainstream effect and lifts off. This interaction with the mainstream increase mixing in the region close to the stagnation point resulting in an increase in effectiveness. When the airfoil is placed further downstream of the injection point (Configuration III-a) the coolant jet does not face any interference flowing forward without having to be redirected. If the local effectiveness pictures are observed, the leading edge of the airfoil is cooled better for higher blowing ratios, this indicates that the momentum of the jet is able to overcome the pressure gradient created by the presence of the airfoil. This is more evident in Figure 64 where the jet streamlines are much closer to the airfoil for higher blowing ratios.

**Wake Effects**

The incoming flow characteristics have a direct effect on endwall film cooling effectiveness. The wake generated by the adjoining walls of the transition pieces enhance mixing of the jet with the mainstream. The cooling capability may quickly be lost preventing the surfaces from being cooled adequately. To understand the effect of wake on film effectiveness, a rectangular plate was placed upstream of the injection point. The distance was varied and temperature distribution was obtained for various blowing ratios. This section describes the experimental readings and computational details specific to these configurations.
Similarly to the other configurations, the measured variables were the $T_c$, the temperature of the coolant inside the plenum; $P_c$ the static pressure of the coolant inside the plenum; and $P_{\text{stat}}$, the main flow static pressure, at the test section. Tests were carried out for four blowing ratios, $M=0.5$, 0.75, 1.0, and 1.5. The test conditions are described in Tables 8 and 9.
Simulation Details

Two different cases were modeled in CFD: (1) Configuration 0-b and (2) 0-c. The grid size ranged from 1,480,380 to 1,710,380 hexahedral cells. The side and top walls were defined as symmetry plane where as a pressure boundary condition was prescribed at the inlet and outlet of mainstream. On the plenum, a mass flow inlet was calculated based on the pressure reading from the experimental runs. The numerical simulation was carried out for the lowest and highest blowing ratios (M=0.5 and 1.5). The values used in the boundary conditions match the experimental measurements listed in Tables 8 and 9.
Table 8: Experimental Measurements for Configuration 0-b

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Mainstream Conditions</th>
<th>Plenum Conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>M = 0.5</td>
<td></td>
<td></td>
</tr>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>339</td>
<td>258</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>101344</td>
</tr>
<tr>
<td>M = 0.75</td>
<td></td>
<td></td>
</tr>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>339</td>
<td>259</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>102009</td>
</tr>
<tr>
<td>M = 1.0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>339</td>
<td>259</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>103096</td>
</tr>
<tr>
<td>M = 1.5</td>
<td></td>
<td></td>
</tr>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>339</td>
<td>259</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>105078</td>
</tr>
</tbody>
</table>
Table 9: Experimental Measurements for Configuration 0-c

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Mainstream Conditions</th>
<th>Plenum Conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>M = 0.5</td>
<td></td>
<td></td>
</tr>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>339</td>
<td>258</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>101346</td>
</tr>
<tr>
<td>M = 0.75</td>
<td></td>
<td></td>
</tr>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>339</td>
<td>260</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>102019</td>
</tr>
<tr>
<td>M = 1.0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>339</td>
<td>261</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>103098</td>
</tr>
<tr>
<td>M = 1.5</td>
<td></td>
<td></td>
</tr>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>339</td>
<td>261</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>105083</td>
</tr>
</tbody>
</table>
Figure 67: Wake effects CFD grid

**Pressure Inlet**
- from measured Velocity and Turbulence

Symmetry Planes

**Mass Flow Rate**
- Nitrogen modeled as “Air”

**Pressure Outlet**
- Static Pressure at the exit

Figure 68: CFD Domain and Boundary Condition Type for Configurations 0-b and c
Results and Discussion

In real engine conditions, endwall film cooling is also affected by the wake induced by the adjacent walls of the transition pieces. Figures 69 and 70 plot average film cooling effectiveness in the presence of wake for test configuration 0-b and 0-c. The presence of wake promotes the mixing of the jet with the mainstream resulting in lower effectiveness. For the cases where the wake plate is placed at x/D=-50.8 upstream of the injection point, the average effectiveness is higher at small x/D. These figures show that for low blowing ratio, the presence of wake increases effectiveness for low x/D. On the contrary, at high x/D effectiveness is higher for the highest blowing ratio.

Figure 69: Average film cooling effectiveness for Configuration 0-b
Figure 70: Average film cooling effectiveness for Configuration 0-c

Figure 71: Local Film Cooling Effectiveness for Configuration 0-b (M=0.5 and 1.5)

Figure 72: Average Film Effectiveness for Configurations 0-c (M=0.5 and 1.5)
As shown in Figures 74 and 75 the numerical simulation aid in the understanding of the vortex formation physics caused by the induced wake. The velocity deficit created by the wake forms a pair of vortices offset from the wake centerline. These vortices lift the jet off the wall promoting the interaction of the jet with the mainstream resulting in a lower effectiveness. The jet interaction with the mainstream causes the jet to lose its cooling capabilities more rapidly which leads to a more sudden decay in film effectiveness. Overall, an increase in the strength of the wake results in a reduction of film effectiveness due to the interaction of the jet with the mainstream flow. The simulation also shows that the wake plate causes the flow to separate forming a recirculation region at the trailing edge of the plate. This recirculation pulls some of the jet flow in the reverse direction. This means that a fraction of the flow is not utilized for the purpose of cooling the region downstream of the injection point as it is intended by design.

![Figure 73: CFD Vector plot for varying x/D – Wake plate](image)

(a) x/D=10  
(b) x/D=20  
(c) x/D=30
Under real engine conditions, the effectiveness of the film has to be evaluated in the presence of both the wake and pressure gradient effects. This section presents the results of the study of combined effect of wake and stagnation region on film cooling effectiveness. Experimental and computational results are presented herein.
Figure 76: Test Configurations for study on stagnation region and wake effects

<table>
<thead>
<tr>
<th>Nomenclature</th>
<th>Distance from Airfoil to LE (x/d)</th>
<th>Distance from Wake Plate to LE (x/d)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1-b</td>
<td>6.4</td>
<td>12.7</td>
</tr>
<tr>
<td>1I-b</td>
<td>12.7</td>
<td>12.7</td>
</tr>
<tr>
<td>1II-b</td>
<td>25.4</td>
<td>12.7</td>
</tr>
<tr>
<td>1-c</td>
<td>6.4</td>
<td>50.8</td>
</tr>
<tr>
<td>1I-c</td>
<td>12.7</td>
<td>50.8</td>
</tr>
<tr>
<td>1II-c</td>
<td>25.4</td>
<td>50.8</td>
</tr>
</tbody>
</table>

Experimental Details

For configurations I-III-b and c, the measured variables were the $T_c$, the temperature of the coolant inside the plenum; $P_c$ the static pressure of the coolant inside the plenum; and $P_{stat}$, the main flow static pressure, at the test section. Tests were carried out for four blowing ratios, $M=0.5, 0.75, 1.0, and 1.5$. The test conditions were as followed:
### Table 11: Experimental Measurements for Configuration I-b

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Mainstream Conditions</th>
<th>Plenum Conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>M = 0.5</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>339</td>
<td>257</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>101329</td>
</tr>
<tr>
<td><strong>M = 0.75</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>339</td>
<td>255</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>102113</td>
</tr>
<tr>
<td><strong>M = 1.0</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>339</td>
<td>260</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>103251</td>
</tr>
<tr>
<td><strong>M = 1.5</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>339</td>
<td>256</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>105457</td>
</tr>
<tr>
<td>Parameter</td>
<td>Mainstream Conditions</td>
<td>Plenum Conditions</td>
</tr>
<tr>
<td>-------------------------------</td>
<td>-----------------------</td>
<td>-------------------</td>
</tr>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>339</td>
<td>258</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>101332</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>M = 0.5</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>339</td>
<td>259</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>102058</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>M = 0.75</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>339</td>
<td>259</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>103073</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>M = 1.0</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>339</td>
<td>259</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>105104</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>M = 1.5</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>339</td>
<td>258</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>105104</td>
</tr>
<tr>
<td>Parameter</td>
<td>Mainstream Conditions</td>
<td>Plenum Conditions</td>
</tr>
<tr>
<td>---------------------------------</td>
<td>-----------------------</td>
<td>-------------------</td>
</tr>
<tr>
<td><strong>M = 0.5</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>339</td>
<td>257</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>101354</td>
</tr>
<tr>
<td><strong>M = 0.75</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>339</td>
<td>255</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>102095</td>
</tr>
<tr>
<td><strong>M = 1.0</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>339</td>
<td>260</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>103251</td>
</tr>
<tr>
<td><strong>M = 1.5</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>P static (Pa)</td>
<td>101118</td>
<td>-</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>339</td>
<td>256</td>
</tr>
<tr>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
<td>105457</td>
</tr>
<tr>
<td></td>
<td>Parameter</td>
<td>Mainstream Conditions</td>
</tr>
<tr>
<td>--------</td>
<td>-------------------------</td>
<td>-----------------------</td>
</tr>
<tr>
<td><strong>M = 0.5</strong></td>
<td>P static (Pa)</td>
<td>101118</td>
</tr>
<tr>
<td></td>
<td>Temperature (K)</td>
<td>339</td>
</tr>
<tr>
<td></td>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
</tr>
<tr>
<td><strong>M = 0.75</strong></td>
<td>P static (Pa)</td>
<td>101118</td>
</tr>
<tr>
<td></td>
<td>Temperature (K)</td>
<td>339</td>
</tr>
<tr>
<td></td>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
</tr>
<tr>
<td><strong>M = 1.0</strong></td>
<td>P static (Pa)</td>
<td>101118</td>
</tr>
<tr>
<td></td>
<td>Temperature (K)</td>
<td>339</td>
</tr>
<tr>
<td></td>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
</tr>
<tr>
<td><strong>M = 1.5</strong></td>
<td>P static (Pa)</td>
<td>101118</td>
</tr>
<tr>
<td></td>
<td>Temperature (K)</td>
<td>339</td>
</tr>
<tr>
<td></td>
<td>Static Plenum Pressure (Pa)</td>
<td>-</td>
</tr>
</tbody>
</table>
Simulation Details

Configurations I-b, III-b, I-c and III-c were modeled. The fluid domains ranged from 1,351,265 to 1,513,265 hexahedral cells. The side and top walls were defined as symmetry plane where as a pressure boundary condition was prescribed at the inlet and outlet of mainstream. On the plenum, a mass flow inlet was calculated based on the pressure reading from the experimental runs. The numerical simulation was carried out for the lowest and highest blowing ratios (M=0.5 and 1.5). The values used in the boundary conditions match the experimental measurements listed in the previous section.

Figure 77: CFD Grid (Configuration III-c)
Results and Discussion

The results presented for these configurations show that the combination of wake and stagnation region results in a decrease of average effectiveness. The presence of wake promotes the mixing of the jet with the mainstream resulting in lower effectiveness. For the cases where the wake plate is placed at x/D=-50.8 upstream of the injection point (Configurations I-c and III-
c), the average effectiveness is higher at small $x/D$. At this location the wake effect is almost negligible and effectiveness values approximate results to Configurations I-a and III-a.

As observed in Figure 91, the proximity of the wake plate results in a recirculation upstream of the injection point. This recirculation region draws coolant air back preventing it from flowing downstream. As the blowing ratio increases the jet has the momentum to overcome these effects and less air is drawn backwards. The jet interaction with the mainstream causes the jet to lose its cooling capabilities more rapidly which leads to a more sudden decay in film effectiveness.

![Figure 79: Average Film Cooling Effectiveness for Configuration 1-b](image-url)
Figure 80: Average Film Cooling Effectiveness for Configuration 1I-b

Figure 81: Average Film Cooling Effectiveness for Configuration 1II-b
Figure 82: Average Film Cooling Effectiveness for Configuration 1-c

Figure 83: Average Film Cooling Effectiveness for Configuration 11-c
Figure 84: Average Film Cooling Effectiveness for Configuration 1II-c

Figure 85: Local Film Cooling Effectiveness for Configuration 1-b

Figure 86: Local Film Cooling Effectiveness for Configuration II-b
Figure 87: Local Film Cooling Effectiveness for Configuration III-b

Figure 88: Local Film Cooling Effectiveness for Configuration I-c

Figure 89: Local Film Cooling Effectiveness for Configuration II-c
The streamline plots presented in Figure 92 shows that the jets are practically undisturbed when the wake plate is placed farther upstream of the injection point. In cases where the wake plate is closer to the injection point, jet flow is altered and mixed with the mainstream flow, particularly for low blowing ratios.
Film Coverage

Higher blowing ratios promote the interaction of the jets with the adjacent jets and with the mainstream flow resulting in a higher effectiveness farther downstream of the injection point. Higher effectiveness promotes jet interaction resulting in higher effectiveness when reattachment of the jet occurs.
Figure 95 shows that film coverage ratio (FCR) is higher at higher blowing ratios. This trend was observed for all configurations. At lower blowing ratios, jet spreading is minimal, making it necessary to implement alternative means of endwall cooling in the region adjacent to the airfoil. The jet to jet interaction is small and hence there is no spreading of the jet. This jet spreading is more evident as the blowing ratio is increased.

![Figure 95: Film Coverage](image)

**Pressure Gradient**

Film cooling effectiveness is also investigated under the presence of mainstream pressure gradient with converging main flow streamlines. The streamwise pressure distribution is attained by placing side inserts into the mainstream. The results are presented for five holes of staggered inclined cylindrical holes. The inclination angle is 30° and the tests were conducted at two Reynolds number, 5000 and 8000.
Experimental Details

Each row of film holes has different pressure ratios because of the inserts along the mainstream. The measured variables were the measured variables were the $T_c$, the temperature of the coolant inside the plenum and mainstream; $P_p$ the static pressure inside the plenum; and $P_{stat}$, the main flow static pressure at the test section.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>$(\rho U)_\alpha$</th>
<th>$(\rho U)_{r,\text{avg}}$</th>
<th>Blowing Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Row 1</td>
<td>25.09</td>
<td>74.36</td>
<td>2.96</td>
</tr>
<tr>
<td>Row 2</td>
<td>30.94</td>
<td>74.36</td>
<td>2.40</td>
</tr>
<tr>
<td>Row 3</td>
<td>40.13</td>
<td>74.36</td>
<td>1.85</td>
</tr>
<tr>
<td>Row 4</td>
<td>56.97</td>
<td>74.36</td>
<td>1.31</td>
</tr>
<tr>
<td>Row 5</td>
<td>93.06</td>
<td>74.36</td>
<td>0.80</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>302</td>
<td>333</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Parameter</th>
<th>$(\rho U)_\alpha$</th>
<th>$(\rho U)_{r}$</th>
<th>Blowing Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Row 1</td>
<td>25.09</td>
<td>131.65</td>
<td>2.96</td>
</tr>
<tr>
<td>Row 2</td>
<td>30.94</td>
<td>131.65</td>
<td>2.40</td>
</tr>
<tr>
<td>Row 3</td>
<td>40.13</td>
<td>131.65</td>
<td>1.85</td>
</tr>
<tr>
<td>Row 4</td>
<td>56.97</td>
<td>131.65</td>
<td>1.31</td>
</tr>
<tr>
<td>Row 5</td>
<td>93.06</td>
<td>131.65</td>
<td>0.80</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>302</td>
<td>333</td>
<td></td>
</tr>
</tbody>
</table>
Simulation Details

Only one configuration was used for this study. The fluid domains contained 1,611,225 hexahedral cells. The side and top walls were defined as symmetry plane where as a pressure boundary condition was prescribed at the outlet of mainstream. At the inlet of mainstream and plenum, a mass flow inlet was calculated prescribed based on readings from the experimental runs. The numerical simulation was carried out for Re=5000 and 8000. The values used in the boundary conditions match the experimental measurements listed in Tables 15 and 16.

Figure 96: Pressure Gradient CFD Grid
Results and Discussion

The average jet Reynolds numbers of 5000 and 8000 are tested and averaged effectiveness is plotted for these cases. Figure 98 shows the values plotted along the mainstream for the $30^\circ$ plate. Experimental data shows that for the first two rows $M$ values is so high that film supply that the jets lift off resulting in very poor coverage and film effectiveness is low. Blowing ratio reduces for rows 3, 4 and 5 the blowing ratio ($M$) and it can be observed an increase in $\eta$ values. Similar trend can be observed in to Re=8000. In addition, the increase in average effectiveness could also be attributed to reattachment of the jet at a higher $x/D$.

The numerical predictions as well as the experimental results report a decrease in effectiveness prior to the following row of holes. This is more pronounced in the numerical
predictions. In part, the discrepancies could be attributed to the turbulence model employed for jet entrainment under accelerating flow conditions. This wave-like behavior could be attributed to the accelerating mainstream which reduces turbulence and constraints jet spreading in the lateral direction. This reduction inhibits the jet to jet interaction and hence jet coverage is less. The effect of mainstream pressure gradient also maintains the film longer. The variation in film cooling effectiveness as a function of x/D is a parameter considered in endwall film cooling hole configurations in order to provide uniform coverage to the metal surfaces.

![Figure 98: Average Film Cooling Effectiveness (Re=5000 and 8000)](image)

The streamlines plots depicted in Figure 100 shows that in the case of higher Reynolds number jet lift off is more evident. Similarly, for both cases, lift off occurs for the first few rows, where blowing ratio is high.
Figure 99: Temperature distribution at Re=5000

Figure 100: Temperature distribution at Re=8000
Local flow field alter the behavior of the coolant flow and its interaction with the mainstream. The trend observed on the numerical effectiveness predictions indicates that there is a local in blowing ratios from hole to hole. The mainstream pressure variation due to the presence of the airfoil, wake and side inserts impact the local characteristics of the flow field resulting in a different blowing ratio that directly impact the behavior of the exiting jet. For low
blowing ratio cases, the data presented in Figure 102 and 103 shows that the blowing ratio average value is higher than the target for configuration 1-b and 0-b. This configuration corresponds to configurations where the wake plate is closer to the injection point. As discussed by Adami et al. [24], wake affects the local features of the coolant mixing and separation of the ejecting flow.
In the presence of mainstream pressure gradient, the row by row blowing ratio variation is significant particularly for the first two rows. High blowing ratio results in jet lift off that result in low effectiveness for x/D close to the injection point. This behavior is similar for the two analyzed cases, Re = 5000 and 8000 as observed in Figures 104 and 105.

Figure 103: Local Blowing Ratios for M = 1.5
Figure 104: Row by Row Blowing Ratio Variation (Re = 5000)

Figure 105: Row by Row Blowing Ratio Variation (Re = 8000)
CHAPTER 5: CONCLUSIONS

The gas turbine industry is continuously under demand to attain higher performance, longer service life and lower emissions. The continuous effort to increase turbine efficiency has promoted the development and sophistication of new material, thermal barriers and cooling methods. Endwall film cooling is a necessity in modern gas turbines for safe and reliable operation.

Performance of endwall film cooling is strongly influenced by among other factors the hot gas flow field. Thus endwall flow field, especially those in utility gas turbines with cannular combustors, is quite complicated in the presence of vortices, wakes and strong favorable pressure gradient with resulting flow acceleration. These flow features can seriously impact film cooling performance and make difficult the prediction of film cooling in endwall.

This study investigated endwall film cooling under the influence of pressure gradient effects due to stagnation region of an axisymmetric airfoil and in mainstream favorable pressure gradient. It also investigates the impact of wake on endwall film cooling near the stagnation region of an airfoil. The investigation consisted of experimental testing and numerical simulation. From the experimental results and numerical predictions the following conclusions can be drawn:

For the Configuration 0-a, film cooling effectiveness is higher for lower blowing ratio at low x/D. At high blowing ratio, the jet lifts off for low x/D, the jet later reattaches effectiveness is increases. High blowing ratio promotes jet to jet interaction resulting in higher jet coverage area.
The presence of an asymmetric airfoil did not change average effectiveness characteristics drastically. Film effectiveness is higher for low x/D at low blowing ratios and at higher x/D effectiveness is higher for high blowing ratios. However, film coverage is impacted by the presence of the airfoil. It was observed that at low blowing ratio, the film flows around the airfoil but is not able to provide coverage in the region adjacent to the airfoil. This may be attributed to the low momentum flux and its inability to overcome the mainstream pressure variations as well as a reduction in lateral spreading of the jet. At lower blowing ratios, jet spreading is minimal, making it necessary to implement alternative means of endwall cooling in the region adjacent to the airfoil.

Wake induced by placing a rectangular plate upstream of the injection point also has an impact on endwall film cooling effectiveness. The cases on which the plate was placed far upstream of the injection point (Configurations 0-c, I-c and III-c) did not affected film cooling effectiveness. The flow was practically undisturbed by the presence of the wake plate. The opposite was observed when the wake plate was close to the injection point (Configurations 0-b, I-b and III-b). The presence of the bluff body formed a recirculation region downstream the trailing edge of the plate drawing coolant flow upstream of the injection point into the recirculation region. This flow intended to cool the region downstream of the holes is now being mixed with the mainstream, loosing its cooling capabilities more rapidly. Turbulence created by the wake, enhanced mixing of the jet flow with the mainstream resulting in a reduction of film cooling effectiveness. The combination of wake and stagnation region reduced film effectiveness furthermore, particularly for cases where the wake plate and airfoil were closest to the injection point. (Configuration 1-b).
The effect of mainstream favorable pressure gradient on film cooling effectiveness was also investigated. Multiple researchers have studied the effect of favorable pressure gradient on airfoil vane cooling but endwall film cooling under accelerating conditions is not fully understood. The numerical predictions as well as the experimental results report a wave-like variation on film cooling effectiveness. This is more pronounced in the numerical analysis. In part, the discrepancies could be attributed to the turbulence model employed for jet entrainment under accelerating flow conditions. However, both results agree that the jet of the first rows of holes lift off due to the variation of blowing ratios as a result of the mainstream pressure distribution. This results in lower effectiveness which is then increased as x/D increases. Several factors could be responsible of the flow behavior. The favorable pressure gradient reduces mainstream turbulence and hence mixing of the jet is reduced leading to lower effectiveness. In addition, it also inhibits lateral jet spreading which also impacts jet coverage.

Last, for the cases of stagnation region and wake effects, numerical analysis aid to the understanding of the interaction of the flow with the mainstream. Flow visualization could serve an additional tool to validate direction of the flow as it enters the mainstream. Also, the impact of the shape of the wake plate trailing edge could also be varied to investigate if film performance can be improved in the region downstream of the bluff body. Much more can be done for the case of mainstream pressure gradient. This configuration can be modeled using various turbulence models to determine which model better predicts film cooling effectiveness under this mainflow conditions. In addition, shaped holes can also be tested and modeled to emulate endwall film cooling hole configurations commonly used in the gas turbine industry. In order for this cooling technique to be effective, film cooling must provide acceptable metal
surface temperatures and reasonable temperature distribution over the endwall to limit thermal stress as a result of large temperature gradients. Better understanding of the behavior and interaction of the jet flow with the mainstream could potentially impact endwall film hole cooling patterns in order to increase endwall film effectiveness.
REFERENCES


