Spring 2015

Benchmarking of Computational Models Against Experimental Data for Adiabatic Film-Cooling Effectiveness for Large Spacing Compound Angle Full Coverage Film Cooling Arrays

Simon R. Martinez

Follow this and additional works at: https://commons.erau.edu/edt

Part of the Aerospace Engineering Commons

Scholarly Commons Citation
https://commons.erau.edu/edt/226

This Thesis - Open Access is brought to you for free and open access by Scholarly Commons. It has been accepted for inclusion in Dissertations and Theses by an authorized administrator of Scholarly Commons. For more information, please contact commons@erau.edu, wolfe309@erau.edu.
Benchmarking of Computational Models Against Experimental Data For Adiabatic Film-Cooling Effectiveness For Large Spacing Compound Angle Full Coverage Film Cooling Arrays

by

SIMON R. MARTINEZ
Embry Riddle Aeronautical University

A Thesis prepared under the direction of the candidate’s committee chairman, Dr. Mark Ricklick, Department of Aerospace Engineering, and has been approved by the members of the thesis committee. It was submitted to the School of Graduate Studies and Research and was accepted in partial fulfillment of the requirements for the degree of Master of Science in Aerospace Engineering.

Spring Term 2015

Thesis Advisor: Professor Mark A. Ricklick
Benchmarking of Computational Models Against Experimental Data For Adiabatic Film-Cooling Effectiveness For Large Spacing Compound Angle Full Coverage Film Cooling Arrays

by

SIMON R. MARTINEZ
Embry Riddle Aeronautical University

A Thesis prepared under the direction of the candidate’s committee chairman, Dr. Mark Ricklick, Department of Aerospace Engineering, and has been approved by the members of the thesis committee. It was submitted to the School of Graduate Studies and Research and was accepted in partial fulfillment of the requirements for the degree of Master of Science in Aerospace Engineering.

THESIS COMMITTEE

[Signatures and signatures]
Chairman, Dr. Mark Ricklick
Member, Dr. Magdy Attia
Member, Dr. Sandra Boetcher

[Signatures]
Department Chair, Dr. Anastasios Lyrintzis or Graduate Program Coordinator, Dr. Yi Zhao
Dean of College of Engineering, Dr. Maj Mirmirani
Associate VP for Academics, Dr. Robert Oxley

[Signatures]
4/1/1/15
Date
4/24/15
Date
4/1/15
Date
ACKNOWLEDGEMENTS

First and foremost I would like thank my mother. Her constant example of work ethic, humbleness, and belief in my dreams, are what built the foundation of what I am today. Without her constant reassurance of my dreams, no matter how out of this world they may have seemed at the time, this path that I am on today would of died long ago in that small house in Rockdale.

I have been blessed with a wonderful family that have motivated me to become the first in my family to earn a Bachelor’s and Master’s degree. For that reason, I would like to thank each and every one of them that have had such a large impact in my life: Grandma Virginia, Grandpa Christopher, Norma, Jaime, Diana, Sarahi, Leonzo, Tony, Jose Sr., Leticia, Esmeralda, Moses, Ernesto, Jesus, Elysia, Remington, Elysia, Joe, Angela, Anabella, Jose Jr., Chris-Angel, Naomi, Melissa, Luke, Tatiana, Eric, Dominic, Christopher, Crystal, Alejandra, Marcos, Andres, Helen, Anali, Ernesto Jr, Oscar, and Sebastian.

I would also like to thank all my close friends and teachers back at home since High School: Lucas, Joseph, Chaston, Alex, Jordan, Mrs.Rowe, and Coach Tooley. Without yall, who knows how lost and boring I would be.

As for my time at Embry-Riddle, I would like to thank all my friends that I have gotten to spend my past 5 years with: John, Stuart, Zach, Omar, Stephen, Mehdi, and Sean. All you guys together have made my college experience an unforgettable one. As for my research team, Royce, Yash, and Bushan thanks for all the help you all gave me along the
way within my research. Without your input and suggestions, the path towards getting this research done would have been much more difficult and boring.

I would also like to give a big thanks to my thesis advisor, Professor Ricklick. Thank you for answering half of my million questions that I had along the process and making me figure out the other half on my own. Also thank you for having high expectations in me and letting me know straight up if I was not up to par to where I needed to be. All those grilling 1-on-1 sessions in your office and all those repetitive practices of presenting my research forced me to get better and thus gave me the confidence to not have any doubt within my abilities during the defense. Also thanks for keeping me fed on a regular basis with all your random fruits while I was in my office all day. Those granny smith apples, bananas, and cuties gave me the energy I needed to tackle those long hours of research in my office.
# TABLE OF CONTENTS

NOMENCLATURE..................................................................................................................... vii
ABSTRACT ................................................................................................................................. viii

CHAPTER 1: INTRODUCTION.....................................................................................................1
  1.1 Overview of Turbine Blade and Component Cooling ....................................................1
  1.2 Film Cooling .....................................................................................................................9
  1.3 Important Parameters Relating To Film Cooling ..........................................................10
  1.4 Extracted Parameters Important To Film Cooling .......................................................14
  1.5 Current Work Objectives .............................................................................................16

CHAPTER 2: LITERATURE REVIEW ......................................................................................18
  2.1 Overview of Turbine Blade Cooling Techniques ...........................................................18
  2.2 Showerhead Cooling ......................................................................................................19
  2.3 Slot Cooling ...................................................................................................................21
  2.4 Full-Coverage Film Cooling .........................................................................................22
    2.4.1 Various Shaped Hole Effects ....................................................................................23
    2.4.2 Full-Coverage Film Cooling: $\alpha$, $\beta$, P/D, X/D and M Effects ........................29
    2.4.3 Turbulence Models Used To Predict Adiabatic Effectiveness .................................29

CHAPTER 3: BENCHMARKING WORK DETAILS ...................................................................32
  3.1 Experimental Study Conducted At University of Central Florida ...............................32
  3.2 Experimental Setup ......................................................................................................33
    3.2.1 Wind Tunnel Setup ..................................................................................................33
    3.2.2 Wind Tunnel Flow Setup Measurements .................................................................34
    3.2.3 Adiabatic Wall Test Surface Setup .........................................................................35
    3.2.4 Gathering Data Through The Use of Temperature Sensitive Paint (TSP) .............36
  3.3 Experimental Results Gathered .....................................................................................36
    3.3.1 Experimental Uncertainty .......................................................................................40

CHAPTER 4: RESULTS ............................................................................................................41
  4.1 Introduction ....................................................................................................................41
  4.2 Computational Domain & Modeling ............................................................................41
  4.3 Boundary Conditions ...................................................................................................48
    4.3.1 Velocity Profile Effects ...........................................................................................50
  4.4 Turbulence Modeling .....................................................................................................55
  4.5 Geometry Comparison with Blowing Ratio ...................................................................63

CHAPTER 5: CONCLUSION ....................................................................................................74

REFERENCES .............................................................................................................................75
LIST OF FIGURES

1-1: Ideal Brayton Cycle ........................................................................................................... 1
1-2: T-S Diagram ....................................................................................................................... 2
1-3: Turbine Blade Cooling Techniques .................................................................................... 4
1-4: Turbine Inlet Temperature Vs. Specific Core Power .......................................................... 5
1-5: Inlet Temperature Improvement Over the Years ................................................................. 6
1-6: Turbine Engine Progress [C = (F-32)/1.8] ....................................................................... 7
1-7: Common Blade Cooling Techniques .................................................................................. 8
1-8: Modern Blade and Vane Using Film Cooling Methods ....................................................... 9
1-9: Liftoff Visualization of Increasing Momentum Flux Ratio .................................................. 13
1-10: Visualization of Parameters Used in Creating Geometry ............................................... 15
1-11: Three-Dimensional View of α and β Orientation .............................................................. 15
2-1: Example of Geometry Used In Slot Cooling; Side Internal View ..................................... 21
2-2: Example of a Continuous Slot Cooling Geometry; Top View ......................................... 22
2-3: Example of a Discrete Slot Cooling Geometry; Top View ............................................... 22
2-4: Visual Thermal Effects of Full Coverage Film Cooling ..................................................... 23
2-5: Shaped Holes Tested in Sargison’s Study ......................................................................... 24
2-6: Schematics of Film Hole Shapes in Hyams Study .............................................................. 26
2-7: Velocity Ratio Distribution for Various Geometries ......................................................... 27
2-8: Temperature Footprints on Test Plate Downstream of Holes ............................................ 28
3-1: Geometry of Wind Tunnel Set Up Within Experimental Study .......................................... 34
3-2: Streamwise Average and RMS Fluctuation Velocity Profiles Measured At Leading Edge of Test Section Using a Constant Temperature Anemometer ........................................................................... 35
3-3: Effects of Varying Blowing Ratios and Lateral Distance between Holes (α = 45 degrees, β = 45 degrees); a. M = 0.4, b. M = 1.6 ................................................................. 38
3-4: Local Adiabatic Film Cooling Effectiveness for FC.C(α = 45 degrees, β = 45 degrees); a. M = 0.4, b. M = 0.8, c. M = 1.6 .................................................................................. 39
3-5: Laterally Averaged Film Cooling Effectiveness; a. FC.A, b. FC.B, c. FC.A RR, d. FC.B RR, e. FC.C, f. FC.D, g. FC.C RR, h. FC.D RR ................................................................. 39
4-1: Domain Numerical Comparison of Cutting Geometry in Half ............................................. 42
4-2: Mesh Independent Study .................................................................................................... 44
4-3: Mesh Geometry with a Cell Count of 58 Million ............................................................... 45
4-4: Close up View of Mesh at Film Hole Locations ............................................................... 45
4-5: Close up Top View of Film Hole Outlet ......................................................................... 46
4-6: All Y+ Wall Treatment Less Than One on Adiabatic Test Surface .................................. 47
4-7: Prism Layers Used Near Adiabatic Wall Location ............................................................. 48
4-8: Far View of Boundary Condition Locations ................................................................. 49
4-9: Close Up View of Boundary Condition Locations .................................................................50
4-10: Velocity Profile Used Within CFD Simulation .........................................................................52
4-11: Effects of Using Velocity Profile Vs Not Using a Velocity Profile ...........................................53
4-12: Effects of Velocity Profile on Thermal Boundary Layer Upstream ...........................................53
4-13: Effects of Velocity Profile on Thermal Boundary Layer Downstream ........................................54
4-14: Global Results Comparison for Each Turbulence Test for FC.C Geometry at M =0.4 ..............56
4-15: All Turbulence Model Testing Against Experimental Data for FC.C M=0.4 ..........................56
4-16: Far Wall Based Turbulence Models Tested Against Experimental Data for FC.C M=0.4 ..........57
4-17: Near Wall Based Turbulence Models Tested Against Experimental Data for FC.C M=0.4 ..........58
4-18: Slope Comparison for Turbulence Models Against Experimental Data for FC.C M=0.4 ..........59
4-19: Comparison of CFD Results of Experimental FC.C M =0.4 ..................................................62
4-20: Comparison of Experimental to CFD Results; FC.A M =0.4 .....................................................66
4-21: Comparison of Experimental to CFD Results; FC.A M=1.6 .......................................................67
4-22: Comparison of Experimental to CFD Results; FC.C M=0.4 .....................................................67
4-23: Comparison of Experimental to CFD Results; FC.C M=0.8 ......................................................68
4-24: Comparison of Experimental to CFD Results; FC.C M=1.6 ......................................................68
4-25: Comparison of Various CFD Results for FC.C at Increasing M ................................................69
4-26: Comparison of Experimental Results for FC.C with Increasing Blowing Ratios M ..................69
4-27: k-ω SST Adiabatic Cooling Effectiveness Distribution for FC.A M=0.4 .....................................70
4-28: k-ω SST Adiabatic Cooling Effectiveness Distribution for FC.C M=0.4 .....................................70
4-29: k-ω SST Adiabatic Cooling Effectiveness Distribution for FC.C M=0.8 .....................................71
4-30: k-ω SST Adiabatic Cooling Effectiveness Distribution for FC.A M=1.6 .....................................71
4-31: k-ω SST Adiabatic Cooling Effectiveness Distribution for FC.C M=1.6 .....................................72
4-32: k-ω SST Adiabatic Cooling Effectiveness Distribution Locally for FC.A M=0.4 .......................72
4-33: k-ω SST Adiabatic Cooling Effectiveness Distribution Locally for FC.C M=0.4 .......................72
4-34: k-ω SST Adiabatic Cooling Effectiveness Distribution Locally for FC.C M=0.8 .......................72
4-35: k-ω SST Adiabatic Cooling Effectiveness Distribution Locally for FC.A M=1.6 .......................73
4-36: k-ω SST Adiabatic Cooling Effectiveness Distribution Locally for FC.C M=1.6 .......................73
LIST OF TABLES

2-1: Factors That Effect Film Cooling Performance ................................................................. 19
2-2: Simulations Tested for Each Geometry in Hyam’s Study .............................................. 25
3-1: Parameters Used In Geometries of Experimental Study .............................................. 32
3-2: Experimental Uncertainty for Various Parameters ......................................................... 40
4-1: Mesh Independent Study Values Gathered ......................................................................... 44
4-2: Mass Flow Variation per Geometry .................................................................................. 49
4-3: Boundary Conditions Used Per Domain Section .............................................................. 50
4-4: Turbulence Test Models Description and Global Results for Geometry FC.C M=0.4 ........ 55
4-5: Geometry and Blowing Ratios Tested for CFD Analysis ................................................. 66
NOMENCLATURE

\[ A = \text{Cross sectional area (m}^2\text{)} \]
\[ U_{\infty} = \text{Freestream Velocity (m/s)} \]
\[ \text{CFD} = \text{Computational Fluid Dynamics} \]
\[ D = \text{Film Hole Diameter (m)} \]
\[ P = \text{Lateral pitch between adjacent holes} \]
\[ X/D = \text{Axial Spacing} \]
\[ h = \text{Heat transfer coefficient (W/m}^2\text{K)} \]
\[ k = \text{Thermal conductivity (W/mK)} \]
\[ L = \text{Length (m)} \]
\[ \dot{m} = \text{Mass flow rate (kg/s)} \]
\[ \alpha = \text{Inclination angle} \]
\[ \beta = \text{Compound angle} \]
\[ \text{Nu} = \text{Nusselt Number} \]
\[ p = \text{Static pressure (Pa)} \]
\[ \Delta p = \text{Pressure drop across test section (Pa)} \]
\[ q'' = \text{Heat flux (W/m}^2\text{)} \]
\[ Re = \text{Reynolds Number} \]
\[ R = \text{Gas constant for air (J/kg*K)} \]
\[ T = \text{Static temperature (K)} \]
\[ T_{\text{aw}} = \text{Adiabatic wall temperature} \]
\[ T_c = \text{Coolant Temperature} \]
\[ M = \text{Blowing ratio} \]
\[ DR = \text{Density Ratio} \]
\[ U = \text{Velocity Magnitude (m/s)} \]
\[ X = \text{Streamwise coordinate (m)} \]
\[ Y = \text{Spanwise coordinate (m)} \]
\[ Z = \text{Vertical coordinate (m)} \]
\[ \gamma = \text{Ratio of specific heats for air} \]
\[ \rho = \text{Density of air (kg/m}^3\text{)} \]
\[ \eta = \text{Adiabatic effectiveness} \]
\[ q'' = \text{Heat flux} \]
\[ RR = \text{Recovery Region} \]
This study aims to benchmark experimental data that tested the effects of blowing ratio, surface angle, and hole spacing for two full coverage geometries composed of cylindrical staggered holes at a compounded angle of 45 degrees. These holes varied parametrically the inclination angle to be 30 and 45 degrees, while maintaining a lateral and axial spacing of 14.5 hole diameters. Within this study, the local film cooling effectiveness was obtained from 30 rows for the 14.5 diameter spacing. Utilizing a velocity profile at the crossflow inlet produced significant differences in the results produced when compared to a uniform freestream velocity profile. The goal of this research is to use commercially available turbulence models, in an effort to provide a benchmarking effect that matches the trusted experimental data found in past experiments. From this study, out of the 7 turbulence model cases tested, it was found that the k-ω Shear Stress Transport turbulence model with no curvature correction provided the best representation of the experimental results being benchmarked.
CHAPTER 1: INTRODUCTION

1.1 Overview of Turbine Blade and Component Cooling

Since the beginning of turbine propulsion technology there has been an understanding that there is a direct correlation that increasing the turbine inlet temperature will directly increase the potential power and efficiency within the system. This ideology came from the ideal Brayton cycle which describes how the thermodynamic workings of a constant pressure engine produces work that can then be extracted from the system. This can be seen in Figure 1-1.

![Figure 1-1: Ideal Brayton Cycle](image)
In regards to engine applications, the location of the engine that witnesses the maximum temperature produced by combustion is the key limiting factor in today’s turbine engines. As can be seen in Figure 1-2, the location between T2 and T3 is where the combustion process occurs in the engine as fuel is burned. As more fuel is burned there becomes a maximum temperature reached, T3. The greater the difference between T3 and T4, the more work that can be extracted from the thermodynamic process. Thus comes the understanding that the higher the T3 reached within the engine, the more energy that can be extracted from the combustion process. However, there is a problem. With today’s fuels, the firing temperature is capable of reaching levels much higher than that of the safe operating ranges of the materials used within the combustion chamber. The typical super alloys used within the turbine engine cannot handle temperatures this high since
their safe operating temperature is roughly under 1500°C while the firing temperature in many of today’s turbine engines exceed 1700°C (Koff, 2004). For this reason the maximum temperature T3 has to be maintained under the safe operating temperature of the components in order to ensure that the components do not fail when exposed to these extreme temperatures. This technique of limiting the maximum temperature T3 to be under the melting temperature of the components was how early turbine systems were designed.

Today, thankfully, through the use of modern blade cooling techniques such as those displayed on Figure 1-3, engineers have been able to allow engine components to operate at temperatures beyond their melting points without jeopardizing the lifespan of the component.
From over the past several decades since Von Ohain and Whittle’s first gas turbine engine, advancements in both materials and cooling techniques have provided much improvement in the gas turbine industry. Documented studies showed that within the past 50 years improvements in materials has increased allowable operating firing temperatures by roughly 195°C while the use of developed cooling techniques, such as those displayed in Figure 1-3, improved operating firing temperatures by 525°C or 11°C per year (Boyce, 2006). Overall this shows the rapid progression of turbine operating temperature due to developed cooling techniques is nearly three times faster than that caused by material advancement. As can be seen in Figure 1-4 and Figure 1-5, the improvement of allowable inlet temperatures within turbine engines over the years, thanks to
advancements in material and developed cooling techniques, has directly improved the specific core power extracted over the years.

Figure 1-4: Turbine Inlet Temperature Vs. Specific Core Power (Koff, 2004)
From these improvements in materials and cooling technologies as well as manufacturing techniques over the years, there has been the creation of various commonly used turbine engines in today’s industries all with growing temperature capabilities as can be seen in Figure 1-6 (Koff, 2004).

Figure 1-5: Inlet Temperature Improvement over the Years (Clifford, 1985)
However, despite all the inlet temperature improvements made over the years, it is important to understand that the air that is used for cooling these turbine components were bled from the relatively cooler fluid from the compressor that is typically a total temperature of approximately 800K within today's engines (Kodzwa & Eaton, 2005). By extracting coolant from the compressor this reduces the efficiency of the machine due to work being required to pump the coolant into the film holes as well as the actual coolant cooling the ambient temperature surrounding the heated components. For this reason it is imperative to make sure that the cooling techniques used within the turbine engine are not only effective in cooling the engine but also efficient so that the minimal amount of coolant is used.

Within today's turbine engines, due to the high combustion temperatures, various components that are exposed to the hot gas require some sort of thermal protection such as by using advanced materials or cooling techniques. Some of these engine components requiring
thermal protection include the stators, blades, endwalls, and the combustor walls. Figure 1-7 and Figure 1-8 displays some of the numerous cooling techniques that are common practice in today’s turbine engines. Impingement cooling has been used to cool the leading edge region of blades as well as the mid-core sections of the blade (Ricklick, 2006). Internal cooling is usually done through pin fins in the trailing edge and ribs or dimples in the mid-cord and leading edges to help promote greater heat transfer between the blade and the cooling fluid. Finally, showerhead and film cooling techniques use cooled air to be shot out at designated locations to protect the blade from the hot ambient surroundings.

![Diagram of blade cooling techniques](image)

**Figure 1-7: Common Blade Cooling Techniques** (Gladden, 1988)
1.2 Film Cooling

Film cooling is characterized by the use of ejecting coolant at discrete locations along a surface that is exposed to a high temperature environment. The goal for this ejected coolant is to
provide a thermal blanket that protects the surface from the surrounding extreme temperatures both locally at the injection location as well as downstream.

Ever since the infancy of turbine blade film cooling during the 1970’s, researchers and engineers quickly found out there was large efficiency boosting rewards within this field (Bogard, 2006). However, with the discover of great potential in this field also came the quick understanding that there was an even greater complexity that needed to be first understood before taking full advantage these rewards. For this reason there has been thousands of research articles written over the past several decades aiming to understand more about every possible aspect of this promising technology. These studies have been on the effects on cooling in regards to turbulence intensity of the main flow, freestream boundary layer thickness, density ratio between free stream and coolant stream, momentum flux ratio, mass flux ratio, blow ratio, hole roughness, hole shape, hole blockage, hole manufacturing techniques, hole inclination angle, hole compound angle, hole length, hole spacing, hole inlet conditions, adverse pressure gradients, downstream of a rotating wake, hole exit shaping, hole embedded in trenches, film jet Mach number, various Reynolds numbers, etc. Due to the vast amount of research articles available concerning the film cooling subject, only studies of direct importance will be presented in this work.

1.3 Important Parameters Relating To Film Cooling

**Hole Diameter: D** – The diameter of the film cooling hole. This parameter is generally the length-scale reference value for most film cooling studies. For today’s turbine engines, film hole diameters are typically just a few millimeters wide in order to maintain the structural rigidity of the blades.
Hole Length: $L$ – The length of the film hole measured from the inlet breakout to the exit breakout. For the non-dimensional hole length $L/D$, a $L/D > 6$ allows the entrance flow effects to diminish before exiting into the free stream. This allows more time for the velocity profile to be affected by the wall which helps to reduce the jet's ability to penetrate into the main flow once exiting. An $L/D < 4$ has been shown to increase the ability of the coolant flow to penetrate into the main flow due to the locally high momentum flux (Natsui, 2012). In terms of film cooling, it is desired to reduce the penetration of coolant into the main flow so that lift off can be reduced.

Inclination Angle: $\alpha$ – Angle measured starting from the surface where the film hole inlet occurs to the angle inclined above that surface. Generally this angle ranges between 10-90 degrees for most film cooling setups. This angle has a major effect of producing lift off of the coolant the closer this inclination angle is to 90 degrees. A visual depiction of the unwanted lift-off this inclination angle can produce if too high can be seen in Figure 1-9.

Compound Angle: $\beta$ – This angle, which can vary between $\pm 90$ degrees, is measured relative to the free stream direction when the axis is projected on to the cooled wall. A visual depiction of this angle can be seen in Figure 1-10. This angle is important because any angle greater than zero will promote spreading of the coolant jet across the test section.

Lateral Pitch: $P$ – The lateral distance between one hole exit and an adjacent hole exit. This distance is usually non-dimensionalized with the hole diameter $D$ to create a reference non-dimensional spacing of $P/D$.

Streamwise Pitch: $X$ – The distance in the streamwise direction that separates two rows of holes. This value is usually normalized to be $X/D$. Both $X/D$ and $P/D$ have a direct effect in the amount
of film coolant expressed to the test section because if these values are small then that means more coolant ejecting holes will be packed closer together.

**Blowing Ratio: M** – This parameter describes the ratio of the mass flux between the coolant and the mainstream hot gas. To determine the blowing ratio, the coolant density $\rho_c$, hot gas density $\rho_\infty$, coolant velocity magnitude $U_c$, and the freestream velocity magnitude $U_\infty$ are needed. This can be seen in the Equation 1.1 below.

$$ M = \frac{(\rho U)_c}{(\rho U)_\infty} \quad (1.1) $$

**Momentum Flux Ratio: I** – This parameter is the ratio between the momentum of the coolant to the momentum of the freestream. This parameter, along with the blowing ratio M, is very important in preventing liftoff of the coolant. A visualization of the effects of lift off can be seen in Figure 1-9.

$$ I = \frac{(\rho U^2)_c}{(\rho U^2)_\infty} = \frac{M^2}{DR} \quad (1.2) $$
Density Ratio: DR – The effect of the density ratio is to create a momentum flux ratio for a predetermined blowing ratio M. This is done by first fixing the blowing ratio and then determining the momentum ratio based off of the density ratio. A DR < 1 is usually only used within laboratory testing since the actual density witnessed within a actual engine is much greater than 1. When the DR is less than 1, it will raise the momentum flux ratio I which will affect the dynamics of the cooling jets. This ratio can be seen in the Equation 1.3 below.

\[ DR = \frac{\rho_c}{\rho_\infty} = \frac{T_\infty}{T_c} \]  

(1.3)

Velocity Ratio: VR – This parameter is the ratio of coolant to mainstream velocity and can be created by the rearrangement of M, DR, and I. This is a common parameter used to describe the flow when both the flow streams are the same temperature.

\[ VR = \frac{U_c}{U_\infty} \]  

(1.4)
1.4 Extracted Parameters Important To Film Cooling

**Heat Flux Reduction:** $q''$ – This parameter describes the amount of heat that passes through a given area as a function of the heat transfer coefficient $h$, and the difference in temperature between the surfaces surrounding the area. However, in terms of adiabatic wall, $q''_f = 0$, the wall would reach a certain temperature distribution which will be defined as $T_{aw}$. This equation for heat flux using an adiabatic wall can be seen in Equation 1.5.

$$q''_f = h_f(T_w - T_{aw})$$ (1.5)

However, this temperature $T_{aw}$ is dependent on the coolant temperature that is injected into the free stream. In order to free from this dependency of the coolant temperature, another parameter, known as adiabatic cooling effectiveness $\eta$, is used.

**Adiabatic Effectiveness:** $\eta$ – This parameter is determined by the ratio of the difference between $T_\infty$ and $T_{aw}$ as well as the difference between $T_\infty$ and $T_c$. This parameter is useful because it no longer has the dependency of having a heat transfer coefficient that depends on the injected coolant temperature. The effectiveness is zero when $T_{aw} = T_\infty$ and is equal to one when $T_{aw} = T_c$. The higher the effectiveness value is to one, the greater the cooling effect the coolant has on the test section. This is due to the fact that if the adiabatic wall is the same temperature of the coolant then that means the coolant was 100% effective in cooling the wall to its temperature. In regards to real turbine engine applications, the adiabatic effectiveness gives a representation of how the local driving temperature will be on the surface of the turbine blade. This will exposed locations where heat transfer will be greatest once conduction is involved within the real turbine engine application.

$$\eta = \frac{T_\infty - T_{aw}}{T_\infty - T_c}$$ (1.6)
Figure 1-10: Visualization of Parameters Used in Creating Geometry

Figure 1-11: Three-Dimensional View of $\alpha$ and $\beta$ Orientation (Natsui, 2010)
1.5 Current Work Objectives

This current work is defined by several objectives all aimed to provide a better understand of film cooling prediction capabilities using commercially available tools. Computation fluid dynamics (CFD) since its infancy has been a very useful tool for engineers to provide predictions of flow characteristics within various applications. However CFD, although a widely used tool, cannot be fully trusted with today’s computational methods which forces companies to spend extra money for corresponding experimental and operational test data. If CFD can gain a greater trustworthiness for its results, it would allow engineers to have greater accuracy of what the final design should be within the preliminary phase. This will reduce the amount of iterations a company needs to have going from the preliminary design phase to the build and test phases. For this reason, comes the need to benchmark against experimental data the various CFD parameters that could cause variation in the solutions. These parameters in question that must be tested in order to provide a reliable computational analysis include but are not limited to mesh size, domain simplifications, turbulence models, and boundary conditions. Once benchmarked, CFD has the capability of providing to the designer the ability to gather various parameter data at specific locations within the test set up with a level of detail that would be very difficult to gather experimentally. This will allow the designer to make accurate adjustments to key areas within the component to create the desired effect.

Within this current work, the experimental data that the CFD will be benchmarked against is that gathered from the University of Central Florida (Natsui, 2012). The CFD software used will be STAR-CCM+ 9.02.007 created by CD-Adapco.
With a thorough understanding of how the various geometries tested experimental at the University of Central Florida compare to the numerical solution provided by CFD, this study intends to make some conclusions on the reliability of various CFD turbulence models and the effects of hole spacing, hole compound angle, hole inclination angle, and blow ratios in regards to cooling effectiveness for cylindrical shaped film holes. Thus, the outcome of this research will help provide additional knowledge to the currently limited topic concerning full coverage film cooling arrays and aims to provide guidance for more accurate applications of CFD within this field.
CHAPTER 2: LITERATURE REVIEW

2.1 Overview of Turbine Blade Cooling Techniques

Over the years there have been various methods developed regarding using coolant to protect the turbine blade from the ambient temperature. Some of the more commonly researched techniques include slot cooling, discrete film cooling, showerhead cooling, and the current technique of this study – full coverage film cooling. Each of these methods have various variables that must be understood in order to provide the best combination that could promote the most cooling effect of the turbine blade. Generally, each of these methods is studied to understand the best combination of the amount of coolant to be used, how the coolant is used, and where the coolant is used. A table of key factors that can affect the performance of film cooling methods can be seen in Table 2-1.

Table 2-1: Factors That Effect Film Cooling Performance (Bogard, 2006)

<table>
<thead>
<tr>
<th>Coolant/Mainstream Conditions</th>
<th>Hole Geometry and Configuration</th>
<th>Airfoil Geometry</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass flux ratio*</td>
<td>Shape of the hole*</td>
<td>Hole location</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- leading edge</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- main body</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- blade tip</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- endwall</td>
</tr>
<tr>
<td>Momentum flux ratio*</td>
<td>Injection angle and compound angle of the coolant hole*</td>
<td></td>
</tr>
<tr>
<td>Mainstream turbulence*</td>
<td>Spacing between holes, P/d</td>
<td></td>
</tr>
<tr>
<td>Coolant density ratio</td>
<td>Length of the hole, l/d</td>
<td>Surface curvature*</td>
</tr>
<tr>
<td>Approach boundary layer</td>
<td>Spacing between rows of holes and number of rows</td>
<td></td>
</tr>
<tr>
<td>Mainstream Mach number</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Unsteady mainstream flow</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rotation</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

* Factors that have a significant effect on predictability of film cooling performance.
2.2 Showerhead Cooling

Showerhead cooling uses generally five to seven rows of closely spaced film holes to inject coolant at the leading edge of a turbine blade or vane. At this typical stagnation location, the structure is subjected to the highest temperatures which make this region the most susceptible to failure due to heating. For this reason high density film coolant is dumped throughout the leading edge to provide a thermal barrier for the structure. A visualization of this cooling method can be seen in Figure 1-3.

In an experimental study conducted by Nathan, the overall adiabatic film effectiveness and the overall cooling effectiveness was measured on a turbine vane that was constructed with materials that would match the Biot number $Bi$ for realistic engine conditions (Nathan & Dyson, 2014). This showerhead blade design consisted of five rows of holes at the leading edge with an additional row of holes on both the pressure and suction regions of the vane. In this study, multiple momentum flux ratios were tested ranging from 0.76 to 6.70. A result of this study helped display particular hot spots in the geometry which helped express the importance in having quality coverage for the adiabatic film effectiveness. Concerning the laterally average overall cooling effectiveness of the geometry, it was found that there was an increase in overall cooling effectiveness as the momentum flux ratio increased. Interestingly, it was found that for the tested momentum flux ratios greater than 4.7, the coolant jets along the radial direction began to merge with the spanwise adjacent coolant jets. This prevented the mainstream flow from penetrating between the coolant holes because a completely blocked barrier was created when the coolant jets merged. This resulted in a sharp increase in adiabatic film effectiveness.
In Polanka’s experimental study, an infrared thermography camera was used to help provide a higher resolution of adiabatic film effectiveness of the showerhead region of the turbine vane model (Polanka, 1999). The goal of this study was to study the effects of blowing ratios (ranging from 0.3 to 2.9) and mainstream turbulence levels (0.5% or 22%) in regards to the pressure surface of a showerhead turbine stator vane. The leading edge of the showerhead geometry used consisted of a row of holes at the stagnation point of the vane. In addition, two additional rows of film cooling holes where placed on the pressure surface of the vane and three additional rows on the suction side. The test was performed in a way so that the approaching stagnation line was positioned either directly in front of the stagnation row of holes of slightly positioned to the suction side of the row. The results of this study displayed increasing effectiveness values in the stagnation region when high turbulence flow was used in higher blowing ratios. This was found to be due to the fact that in higher mainstream turbulent flows, it helps spread the coolant evenly throughout the stagnation region. However, this improvement in effectiveness was only found to occur near the stagnation region for high turbulent flows, because downstream the effectiveness reduced for all other blowing ratios.

In Witteveld’s study (Witteveld, 1999), it complemented the study conducted by Polanka (Polanka, 1999) stated above in the fact that the same experimental parameters where used. However, in Polanka’s study it studied the near pressure side of the showerhead cooling vane while in Witteveld’s study it focused on the near suction side of the vane. In this study it found that for the near suction side of the vane, when at low blowing ratios, high turbulence values produced the greatest cooling effectiveness for the showerhead region. However, for high blowing ratios, the lower turbulence case produced the greatest effectiveness for the region. Additionally, again it was
found that with a high enough blowing ratio, a spanwise merging of adjacent cooling jets would occur producing a protective barrier from the mainstream flow. Once this phenomenon occurred, it created a jump in adiabatic effectiveness for the region under the protective barrier.

In Cutbirth’s and Bogard’s study, a showerhead consisting of six rows of spanwise coolant holes was tested in blowing ratios ranging from 0.5 to 2.0 in both high and low turbulence levels (Cutbirth & Bogard). Using an infrared camera to capture the flow interaction at the stagnation region of the vane, the results were similar to that found in Nathan’s work were it was found that a thermal barrier was created with laterally adjacent holes for higher blow ratios (Nathan & Dyson, 2014). This thus resulted in preventing the mainstream flow from interacting with the coolant holes and thus promoted a higher adiabatic effectiveness throughout the stagnation region of the turbine vane.

2.3 Slot Cooling

The approach of slot cooling is using a two dimensional slot to inject coolant film over the blade to produce a cooling effect over the test section. An example of geometries used for releasing coolant out of this 2-D slot can be seen in Figure 2-1, Figure 2-2 and Figure 2-3.

![Figure 2-1: Example of Geometry Used In Slot Cooling; Side Internal View](image)
In Bruce-Black’s study of various slot cooling geometries, it was discovered that when decreasing the width of the slot it led to a substantial increase in adiabatic effectiveness (Bruce-Black, 2011). In addition, it was found that a continuous slot geometry had a much higher adiabatic effectiveness over the test section then a discrete slot in every condition tested. However, in this study, it was discussed that in practical engine applications the discrete slot geometry would be used over the continuous slot due to the added structural integrity the discrete slot geometry provides.

2.4 Full-Coverage Film Cooling

Full-coverage film cooling is the use of various coolant ejecting holes to cover a test section area in order to thermally protect it from the surrounding freestream temperature. Full
coverage film cooling has the ability to cool not only the immediate region where the coolant is being ejected but also in the downstream region. A visual description on the cooling effects of film-cooling over a surface can be seen in Figure 2-4.

![Figure 2-4: Visual Thermal Effects of Full Coverage Film Cooling (Natsui, 2012)](image)

2.4.1 Various Shaped Hole Effects

An experimental study by Sargison tested the cooling performance of cylindrical (Sargison, 2002), fan-shaped, slot- and console hole configurations as can be seen in Figure 2-5. It was found that the console shaped hole produced the least aerodynamic losses of all the shapes concerning exit pressure measured of the test site. In addition, the cylindrical shaped holes were found to have the highest calculated heat transfer coefficient of all the shapes in every lateral spacing variation tested. This is a detrimental parameter to be high in for it promotes more heat to pass into the testing sight which makes it more difficult for the coolant to cool the testing surface.
Thus, in regards to wanting to minimize the heat transfer coefficient, the fan shaped hole produced the lowest heat transfer coefficient values for every lateral spacing variation tested. In addition, in regards to wanting to create the highest effectiveness possible, the slot, fan shaped, and console shaped holes all performed better than the cylindrical hole for all lateral spacing variations tested.

A computational study by Hyams tested different shaped holes at blowing ratios of 1.25 and 1.88 in order to study the adiabatic effectiveness capabilities of these geometries. Figure 2-6 shows the geometries and Table 2-2 shows the simulations tested for these shapes concerning density ratios and blowing ratios. From this study, as can be seen in Figure 2-7, it showed that the cylindrical hole had the highest average velocity ratio while the laterally-diffused hole had the least. This expressed the impacts of these exiting velocity ratios because although the same mass flow of coolant is exiting each hole, since the laterally-diffused hole has a bigger exiting area, the velocity of the coolant exiting is less and thus so is the momentum. This reduction in momentum exiting the hole allows more prevention of lift-off which thus helps provide a more effective cooling effect to the turbine blade. The cooling effect of each of these holes can be seen in Figure 2-8 where it shows the variation of temperature downstream of each hole geometry. Due to the
laterally-diffused hole geometry’s capability to prevent lift off as well as cover a larger area of the turbine blade with coolant, this resulted in the laterally-diffused hole producing the best adiabatic effectiveness capabilities of all the geometries tested within the scope of this study.

Table 2-2: Simulations Tested for Each Geometry in Hyam’s Study (Hyams, 2000)

<table>
<thead>
<tr>
<th>Film Hole Shape</th>
<th>M</th>
<th>DR</th>
<th>test surface</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cylindrical (REF)</td>
<td>1.25, 1.88</td>
<td>1.6</td>
<td>adiabatic, const. q&quot;</td>
</tr>
<tr>
<td>Forward Diffused (FDIFF)</td>
<td>1.25, 1.88</td>
<td>1.6</td>
<td>adiabatic, const. q&quot;</td>
</tr>
<tr>
<td>Laterally Diffused (LDIFF)</td>
<td>1.25, 1.88</td>
<td>1.6</td>
<td>adiabatic, const. q&quot;</td>
</tr>
<tr>
<td>Inlet Shaped (ISHAP)</td>
<td>1.25, 1.88</td>
<td>1.6</td>
<td>adiabatic, const. q&quot;</td>
</tr>
<tr>
<td>Cusp Shaped (CUSP)</td>
<td>1.25</td>
<td>1.08</td>
<td>adiabatic, const. q&quot;</td>
</tr>
</tbody>
</table>
Figure 2-6: Schematics of Film Hole Shapes in Hyams Study (Hyams, 2000)
Figure 2-7: Velocity Ratio Distribution for Various Geometries (Hyams, 2000)
Figure 2-8: Temperature Footprints on Test Plate Downstream of Holes (Hyams, 2000)
2.4.2 Full-Coverage Film Cooling: $\alpha$, $\beta$, P/D, X/D and M Effects

In an experimental study conducted by Mayle, the adiabatic effectiveness and heat transfer measurements were tested for cylindrical film holes that were at an inclination angle of 30 degrees and a compound angle of 45 degrees (Mayle, 1974). In addition, these geometries were tested at various spanwise spacing P/D of 8, 10, and 14 for a blowing ratio M of 0.5, 1.0, 1.5, and 2.0. Within this experiment is was found that there was significantly higher effectiveness values obtained for geometries with lower spanwise spacing P/D, regardless of the blowing ratio. In addition, it was noticed that the maximum effectiveness was obtained for the blowing ration of M =1.5. This was mainly due to the fact that for the M =2.0, there began to have significant liftoff of the coolant from the test surface which prevented the coolant from interacting with the surface effectively. An example of this “lift off” phenomena can be seen in Figure 1-9. In addition, the compound angle $\beta$ was found to be helpful in spreading the coolant in the lateral direction which helped cover more surface with coolant.

2.4.3 Turbulence Models Used To Predict Adiabatic Effectiveness

In a computational study conducted by Mr. Deepak Raj P.Y, the effects of blowing ratios (M = 0.6 to M = 2.4) on a flat plate using combined impingement jets and film cooling jets was conducted (Raj P.Y., 2013). In this test, the calculations were carried out by using the k-$\omega$ shear stress transport and the k-$\varepsilon$ standard turbulence model. From the results of this study when benchmarking against experimental data, it was found that the k-$\omega$ shear stress transport turbulence model produced the best representation of the experimental data. This turbulence model produced a solution that over predicted the experimental data surface total temperature by roughly 8%. If converted in terms of adiabatic effectiveness, this would result in a slight under prediction of the
effectiveness. Thus, this turbulence model displayed adequate capabilities when simulating thermodynamic problems regarding film cooling for these blowing ratios.

In a computational study by Walters, it aimed to use validated test cases of a row of discrete jets within a flat plate structure in order to test the accuracy of various computational techniques (such as proper modeling of flow physics, proper grid generation, higher-order discretization scheme, and effective turbulence modeling) in regards to predicting film cooling results (Walters & Leylek, 1997). The various flat plate geometries that were validated against had holes with a streamwise injection angle of 35 degree, two film hole stream wise spacing’s of $X/D = 3.5$ and $X/D = 1.75$, blowing ratios varied from 0.5 to 2.0, free stream turbulence intensity of 2 percent, and a density ratio of 2.0. From the literature review stated in this study, it described how when using computational methods for cases where film holes have “realistic” (short) spacing’s that the flow field begins to exhibit a three-way coupling interaction. This interaction behavior occurs between the crossflow, film-hole, and the plenum regions, which shown from past literature, often causes problems if there is an inadequate grid geometry produced that is not fine enough at these mixing locations to capture this phenomenon. Thus, in Walters study, through the use of fully elliptic Reynolds-averaged Navier-Stokes computations, it was shown that this turbulence model was unable to predict the behavior accurately of the jet lift off experience within the experimental data for any of the blowing ratios tested. This limitation for this turbulence model to capture the lift off flow behaviors displayed how additional considerations for different turbulence models should be used when simulating this kind of interaction. It was suggested that possible near-wall treatments, two-layer, or low-Re turbulence models could be used to provide a better prediction of the flow near the jet exits (Walters & Leylek, 1997).
In another study conducted by Butkiewiez, it discovered that the use of the k-ε turbulence model in their study depicted contrary results by over predicting the reattachment length of a test for two-dimensional normal jets (Butkiewiez, Malters, McGovern, & Leylek, 1995). This is the opposite of what has been found in many past literature studies where k-ε has been often found to under predict reattachment length by nearly 20 percent (Walters & Leylek, 1997).
CHAPTER 3: BENCHMARKING WORK DETAILS

3.1 Experimental Study Conducted At University of Central Florida

The experimental study being benchmarked in this work was conducted at the University of Central Florida by Greg Natsui, Roberto Claretti, and Jayanta S. Kapat in 2012 (Natsui, 2012). This experiment aimed to obtain adiabatic film cooling effectiveness contours for various full coverage geometries using temperature sensitive paint on a low thermal conductivity surface. The effects of blowing ratio, hole spacing, and inclination angle are tested using cylindrical shaped holes. There were a total of four different geometries tested in the experimental study at the University of Central Florida. Table 3-1 gives a detailed description of the parameters for the two geometries used within the current CFD study for benchmark comparison. Within these experimental tested geometries, the inclination angle tested was either 30 or 45 degrees, compound angle stayed a constant 45 degrees, axial and lateral spacing was a nondimensional distance of either 14.5 X/D or 19.8 X/D, and the number of rows used axially for the recovery region was 20 for FC.A, 16 for FC.B, 20 for FC.C, and 15 for FC.D, while for the full coverage testing without the extended recovery region was 30 rows for FC.A, 23 rows for FC.B, 30 rows for FC.C, and 22 rows for FC.D.

Table 3-1: Parameters Used In Geometries of Experimental Study That Were Benchmarked Against In This CFD Study (Natsui, 2012)

<table>
<thead>
<tr>
<th>Geometry</th>
<th>Inclination Angle, α (degrees)</th>
<th>Compound Angle, β (degrees)</th>
<th>Axial Spacing, X/D</th>
<th>Lateral Spacing, P/D</th>
<th>Number of Rows Axially Nₙ (2 plate + RR/3 plate) (with RR/without RR)</th>
</tr>
</thead>
<tbody>
<tr>
<td>FC.A</td>
<td>30</td>
<td>45</td>
<td>14.5</td>
<td>14.5</td>
<td>20/30</td>
</tr>
<tr>
<td>FC.C</td>
<td>45</td>
<td>45</td>
<td>14.5</td>
<td>14.5</td>
<td>20/30</td>
</tr>
</tbody>
</table>
3.2 Experimental Setup

3.2.1 Wind Tunnel Setup

The wind tunnel had to be created in order to work with the 1.2m X 0.55m test section used within this study. At the test section location, there was a cross section of the primary duct that was 15cm in the wall normal direction and 1.1m in the wall lateral direction. When compared to the D = 2.5mm film holes, this corresponded to a height of 60D for the test matrix. The reason for this set up was to ensure that there was no interruption of the coolant jets leaving the holes caused by the duct. In addition, in order to ensure that the main flow velocity is not increased substantially by the creation of the gradient boundary layer from the film holes, the area of the cross flow was made to be 200 times larger than the total area of the film holes. By making the main cross flow this much bigger than the film holes it allowed that for when the maximum blowing ratio of M = 2.0 was tested, it only caused a 1% increase in cross flow velocity at the most downstream row of film holes. This resulted in the main flow velocity upstream to be $U_\infty = 27$ m/s and the increase at the most downstream row of film holes to be 0.3m/s. A general schematic of the wind tunnel set up used can be seen in Figure 3-1.
3.2.2 Wind Tunnel Flow Setup Measurements

Various static pressure readings were conducted along the wind tunnel test section to ensure there was not any detrimental pressure gradients that were created whenever coolant was injected into the mainstream causing a boundary layer to develop (Natsui, 2012).

Using a constant temperature anemometer to measure the freestream velocity and turbulence intensities within the test section it was found that the resulting Reynolds number was $Re_D=4300$. This Reynolds number was calculated from the measured freestream velocity of $U_\infty \approx 27\text{m/s}$ and hole diameter of $D = 2.5\text{mm}$. From the measurements gathered at the leading edge of the test section, the streamwise mean, turbulence intensity, and RMS fluctuation velocity profile was graphed which can be seen in Figure 3-2.
With the data displayed on Figure 3-2, it can be seen that the turbulent fluctuations in the main stream flow died down to $\text{TI} < 1\%$. However, to ensure that a turbulent boundary layer was present at the first row of film cooling holes, the flow was tripped 36D upstream of the test section. The leading edge boundary layer thickness of the test section was found to be $\delta = 8.8\text{mm}$ and displayed no lateral variation (Natsui, 2012).

### 3.2.3 Adiabatic Wall Test Surface Setup

The geometries and their inclination angle $\alpha$ and compound $\beta$ angle parameters can be seen in Table 3-1 and a visual representation on Figure 1-10. Each of these test sections had the same set up for the experiment.

In order to produce as close to an adiabatic wall as possible, the test section surface was covered with 10mm thickness of Rohacell RIMA which is a low density closed cell foam with a thermal conductivity of $k = 0.029\text{W/m-K}$. 

![Figure 3-2: Streamwise Average and RMS Fluctuation Velocity Profiles Measured At Leading Edge of Test Section Using a Constant Temperature Anemometer](image_url)
3.2.4 Gathering Data Through The Use of Temperature Sensitive Paint (TSP)

This experiment extracted temperature measurements from the test section through the use of temperature sensitive paint. The creation of temperature sensitive paint is done by combining a luminescent molecule with a transparent polymer binder. In this experiment TSP was used to cover the surface and a light was shined on it with a 475nm wavelength to excite the molecules. For this paint, the intensity of the light emitted from the paint is inversely proportional to the temperature. Then, the intensity distributions of the paint is captured through the use of a CCD camera and long pass filter. This camera used took images at a rate of 200-350ms in order to capture the fluorescence given off by the paint at a steady state.

3.3 Experimental Results Gathered

Results of the experimental study are displayed on Figure 3-3, Figure 3-4, and Figure 3-5. On Figure 3-3, the effects of varying the lateral spacing between holes, P/D, can be seen for blowing ratios of M = 0.4 and M = 1.6. From this figure, it displays how the average lateral effectiveness is improved as lateral spacing between holes is decreased and blowing ratios are increased.

On Figure 3-4, the local distributions of film cooling effectiveness is displayed for the FC.C geometry. As can be seen from this figure, at the lowest blowing ratio tested, M = 0.4, the jets seem to be well attached to the test section which is shown by the high local values of effectiveness right after the location the coolant was injected (Natsui, 2010). As the blowing ratio is increase, the jets begin to detach which is most shown in M = 1.6. At this high blowing ratio the jets are clearly lifted off the surface at the injection site, but are returned to the surface further downstream and end up providing the highest and most uniform magnitude of cooling. In addition,
for higher blowing ratios, the compound injection angle $\beta$ is more effective in spreading the coolant in the lateral direction to provide better coverage of the coolant. This is due to the fact that at this compound angle, when at higher blowing ratios, there is more lateral momentum to resist the mainstream flow.

In Figure 3-5, it shows the full set of all the laterally averaged effectiveness test done within this experiment. For all geometries it showed that as blowing ratio increased so did the overall laterally average effectiveness. For blowing ratios at $M = 1.6$ the first several rows had a considerably lower adiabatic effectiveness then the lower blowing ratios. This was due to the effects of lift off that occurs with high blowing ratios. However this coolant does return to the surface further downstream to cool the surface of the test section. In addition, the effects of the injection angle seemed to do a second order impact on the laterally averaged effectiveness throughout several rows. However, there were some noticeable changes of cooling capabilities locally near the hole caused from the injection angle which can be better seen in Figure 3-4.
Figure 3-3: Effects of Varying Blowing Ratios and Lateral Distance between Holes ($\alpha = 45$ degrees, $\beta = 45$ degrees); a. $M = 0.4$, b. $M = 1.6$ (Natsui, 2012)
Figure 3-4: Local Adiabatic Film Cooling Effectiveness for FC.C (α = 45 degrees, β = 45 degrees); a. M = 0.4, b. M = 0.8, c. M = 1.6 (Natsui, 2012)

Figure 3-5: Laterally Averaged Film Cooling Effectiveness: a. FC.A, b. FC.B, c. FC.A RR, d. FC.B RR, e. FC.C, f. FC.D, g. FC.C RR, h. FC.D RR (Natsui, 2012)
3.3.1 Experimental Uncertainty

In this experiment the propagation error effect on the absolute error of the resultant, R, was found through the use of Equation 3.1. The uncertainty for various parameters within this experiment is shown in Table 3-2. An important uncertainty parameter is that of blowing ratio and effectiveness being of 10% uncertainty. This uncertainty for blowing ratio and effectiveness will play an important role when discussing the benchmark comparison of CFD data to this experiment in Chapter 4.

\[ uR = \frac{u_c}{u_\infty} \sqrt{\sum_{i=1}^{n} \left( \frac{\partial R}{\partial x_i} \mu_{x_i} \right)^2} \]  

(3.1)

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Units</th>
<th>Typical Value</th>
<th>Total Uncertainty (20:1)</th>
<th>Percent Uncertainty</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coolant Mass Flow</td>
<td>kg/s</td>
<td>0.022</td>
<td>0.00032</td>
<td>1.5%</td>
</tr>
<tr>
<td>Coolant Area</td>
<td>m²</td>
<td>0.0018</td>
<td>0.000087</td>
<td>4.8%</td>
</tr>
<tr>
<td>Mainstream Mass Flux</td>
<td>kg/s/m²</td>
<td>23.85</td>
<td>1.86</td>
<td>7.8%</td>
</tr>
<tr>
<td>Blowing Ratio</td>
<td>-</td>
<td>0.63</td>
<td>0.063</td>
<td>10%</td>
</tr>
<tr>
<td>Effectiveness</td>
<td>-</td>
<td>0.2</td>
<td>0.02</td>
<td>10%</td>
</tr>
</tbody>
</table>
CHAPTER 4: RESULTS

4.1 Introduction

Although experimental data has been used to understand fluid dynamic research for many years, there are still some limitations in the level of detail that can be captured by typical experiments. For this reason, comes the importance of using CFD analysis to provide a benchmark study of past experimental data in order to yield some aspects of the flow that would normally not be easily seen throughout experiments.

As discussed in the literature review, there are many different variables that could provide an effect in the analysis of effectiveness and thus various turbulence models have been tested for various situations in order to provide the most accurate results. However, although it has been shown that with today’s turbulence models that effectiveness is usually under predicted compared to results found in experiments, the overall trend and flow patterns captured by CFD has been shown to be quite well from past research. Thus, the main purpose of this work is provide a numerical analysis in order to support experimental data gathered from a past experiment done at the University of Central Florida (Natsui, 2010).

4.2 Computational Domain & Modeling

In this current work, a computational domain was created to accurately represent two geometries used in experimental study done by Natsui (Natsui, 2010). This included the plenum, exit into the atmosphere, as well as all aerodynamic parameters found in the experimental study.

Because the geometry size was so large with a length and width of 1.2m and 0.55m respectively and the need to have detailed analysis of the film holes that were 2.5mm in diameter,
it became important to find ways to reduce the computational cost of the analysis without losing accuracy. For this reason, since the geometry was symmetrical along the stream wise direction, the geometry was divided in half and tested to verify that no change in the numerical solution occurred. The test results can be seen in Figure 4-1 which display how no visual changes occurred when dividing the symmetrical geometry in half. The results of no change in the solution after dividing the geometry in half was expected since it has been done before in past experiments from the literature (El-Gabry & Kaminski, 2005).

Once it was verified that cutting the domain in half would not change the solution, a mesh independent study was conducted. As can be seen in Table 4-1 and Figure 4-2, the surface average temperature returned from the computer simulation began to level off between a cell count
of 16 million to 58 million with the range between the values being returned only differencing by 0.2 degrees K or 0.063%. From the visual seen in Figure 4-2 of the change in surface average temperature as a function of cell count, initially it was decided to use a mesh size with a cell count of 36 million. However, as later analysis was conducted using the k-ω SST turbulence model at increasing blowing ratios, it was shown that this cell count was not capable of producing a converged solution for the complex flow witnessed for the blowing ratio of M = 0.8 case. For this reason, the final cell count used for the mesh for all the results displayed in this study was 58 million cells which was fine enough to provide a converged solution that captured the flow phenomena during this blowing ratio. A visual of how the final geometry with this cell count looked like can be seen in Figure 4-3, Figure 4-4, and Figure 4-5. Variation in fineness and coarseness of the mesh was implemented at different locations of the mesh to be computationally efficient as well as to maintain accuracy within key locations. As can be seen in Figure 4-4 and Figure 4-5, approximately 40 cells was used to cover the hole diameter D (2.5 mm) at the adiabatic wall in order to capture the flow phenomenon at this location because higher accuracy is desired here and higher variation of values is expected at this location. This is different then what is expected further away from the film jet holes as can be seen Figure 4-3, which is why approximately 100 cells was used to capture flow within 60D height within the cross flow inlet, cross flow outlet, and the overall air duct.
**Table 4-1: Mesh Independent Study Values Gathered**

<table>
<thead>
<tr>
<th>Cell Count (millions)</th>
<th>Surface Average Temperature (K)</th>
<th>Percent Change In Surface Average Temperature</th>
</tr>
</thead>
<tbody>
<tr>
<td>58</td>
<td>318.1</td>
<td>0.0313 %</td>
</tr>
<tr>
<td>36</td>
<td>318.0</td>
<td>0.0313 %</td>
</tr>
<tr>
<td>24</td>
<td>318.1</td>
<td>0.0314 %</td>
</tr>
<tr>
<td>16</td>
<td>318.2</td>
<td>0.25 %</td>
</tr>
<tr>
<td>10</td>
<td>319</td>
<td>N/A</td>
</tr>
</tbody>
</table>

**Figure 4-2: Mesh Independent Study**
Figure 4-3: Mesh Geometry with a Cell Count of 58 Million

Figure 4-4: Close up View of Mesh at Film Hole Locations
In order to ensure to capture the heat transfer and fluid effects near the wall of the adiabatic test section, the all y+ wall treatment was set to be less than 1 throughout the region as can be seen in Figure 4-6. The reason for this all y+ wall treatment is so that the near-wall cells within the boundary layer region can properly produce accurate results. This y+ wall treatment allows the simulation to properly connect the viscosity affected boundary layer region near the wall with the fully turbulent region, which is then calculated through the use of turbulence models.
Near the adiabatic wall test section, in the aim to both capture the presence of the boundary layer near the wall as well as save computational cost, 15 prism layers were used with a 20% increase in size per layer. This allowed a fine enough mesh distribution to be present near the wall region where the boundary layer is expected to be. Figure 4-7 gives a visual representation of the prism layers used within the geometry.
4.3 Boundary Conditions

The boundary conditions for the CFD simulation using Star-CCM+ was set to replicate the same boundary conditions found in the experimental testing done by Natsui.

As for various key areas within the geometry such as the cross flow inlet, plenum inlet, cross flow outlet, adiabatic wall, and the film holes, the boundary conditions placed for each of these locations can be seen in Figures 4-8 and Figure 4-9 as well as Table 4-3. For the cross flow inlet, the temperature was set at 300K while the plenum inlet (coolant location) was set at 400K. In order to mimic the blowing ratio for the different test, the mass flow within the plenum inlet was changed, as can be seen in Table 4-2.
### Table 4-2: Mass Flow Variation per Geometry

<table>
<thead>
<tr>
<th>Blowing Ratio</th>
<th>Geometry</th>
<th>Mass Flow (kg/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.4</td>
<td>FC.A &amp; FC.C</td>
<td>0.00467914</td>
</tr>
<tr>
<td>0.8</td>
<td>FC.C</td>
<td>0.0092052736</td>
</tr>
<tr>
<td>1.6</td>
<td>FC.A &amp; FC.C</td>
<td>0.0184105472</td>
</tr>
</tbody>
</table>

**Figure 4-8: Far View of Boundary Condition Locations**
The boundary conditions regarding the velocity profile was tested within this study using the realizable k-ε turbulence model. The reason for using this turbulence model is because past research has shown accuracy and reliability of this turbulence model for similar full coverage film cooling geometries in the past (Natsui, 2010). For this reason, the realizable k-ε turbulence model was found suitable as the initial turbulence model to use for the study of the effects of

### 4.3.1 Velocity Profile Effects

The boundary conditions regarding the velocity profile was tested within this study using the realizable k-ε turbulence model. The reason for using this turbulence model is because past research has shown accuracy and reliability of this turbulence model for similar full coverage film cooling geometries in the past (Natsui, 2010). For this reason, the realizable k-ε turbulence model was found suitable as the initial turbulence model to use for the study of the effects of

---

**Table 4-3: Boundary Conditions Used Per Domain Section**

<table>
<thead>
<tr>
<th>Section</th>
<th>Boundary Conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cross Flow Inlet</td>
<td>Velocity Profile</td>
</tr>
<tr>
<td>Cross Flow Outlet</td>
<td>Pressure Outlet</td>
</tr>
<tr>
<td>Plenum Inlet</td>
<td>Mass Flow Inlet</td>
</tr>
<tr>
<td>Adiabatic Wall</td>
<td>Adiabatic, No-Slip Condition</td>
</tr>
</tbody>
</table>

---

Figure 4-9: Close Up View of Boundary Condition Locations
implementing a velocity profile. Although the CFD software allows for a uniform velocity inlet profile, with the motivation to replicate the experimental data as much as possible, the effects of including the velocity profile witnessed during the experiment was tested. Using the velocity profile data found on Figure 3-2 that was provided by the experimental study, the velocity profile displayed in Figure 4-10 was implemented as a boundary condition within the cross flow inlet location in the CFD simulation. Once this velocity profile was used, it was shown that there were some distinct differences gained within the analysis when compared to the experimental data. As can be seen in Figure 4-11, from the location of film hole rows 1-6, there is nearly no difference between the effects of using a velocity profile vs not using a velocity profile. This can be attributed to the turbulence model used (Realizable k-ε) within this velocity profile comparison’s capability to replicate the same thermal boundary layer for these two cases for rows 1-6 which can be seen in Figure 4-12. However, for the locations downstream at rows 7-16 the velocity profile case’s thermal boundary layer becomes more pronounced sooner than the non-velocity profile case as can be seen in Figure 4-12. This causes an increase in spanwise average adiabatic effectiveness for the case with the velocity profile as opposed to the non-velocity profile case for rows 7-16 which can be seen in Figure 4-11. For the remainder of the rows 16-30, adiabatic effectiveness is shown to be higher for the non-velocity profile case as can be seen in Figure 4-11. This higher result for adiabatic effectiveness for the non-velocity profile case within far downstream locations was found to be due to the more pronounced thermal boundary layer produced from the simulation as can be seen in Figure 4-13. Because a thermally cooler environment was produced due to the greater thermal boundary layer cooling effect of the non-velocity profile case, it resulted in providing a
surface temperature cooler than that of the velocity profile case which resulted in a higher adiabatic effectiveness value at these downstream locations.

Figure 4-10: Velocity Profile Used Within CFD Simulation
Figure 4-11: Effects of Using Velocity Profile Vs Not Using a Velocity Profile

Figure 4-12: Effects of Velocity Profile on Thermal Boundary Layer Upstream
The results of this velocity profile effect’s study on the CFD simulations displayed that there will be differences in adiabatic effectiveness values produced whether a velocity profile or a uniform flow is used. However, since in the real life experiment that is being used to benchmark this study had a specified velocity profile at the cross flow inlet, as displayed in Figure 4-10, for the remainder of this study to maintain identical representation of the experimental setup, this same velocity profile will be used to describe the flow at the cross flow inlet for the upcoming turbulence modeling comparisons and geometry comparisons.
4.4 Turbulence Modeling

In order to provide the most accurate predictions using CFD analysis, there must be a focus to use the best turbulence model for the given study. For this reason, 7 different variations of turbulence models and features were tested. Each of these turbulence models test can be seen in Table 4-3 and their comparison against the experimental data for FCC geometry at M = 0.4 can be seen in Figure 4-14, Figure 4-15, and Figure 4-16. As can be seen in Figure 4-14, the k-ω SST turbulence model with Curvature Correction option turned off, provided the best adiabatic effectiveness average of the entire geometry when compared at a global scale. In Figure 4-15, again it can be seen that k-ω SST provided again the best prediction of the experimental data when looked at in within the span wise direction average for adiabatic effectiveness. In Figure 4-18 it shows a close up view of how each turbulence test did compared to the experimental in regards to mimicking the slope of decreasing effectiveness immediately after each hole. Again, in regards to the slope prediction, k-ω SST provided the best results in replicating the experimental data.

Table 4-4: Turbulence Test Models Description and Global Results for Geometry FC.C M=0.4

<table>
<thead>
<tr>
<th>Turbulence Test #</th>
<th>Details</th>
<th>Global Average η</th>
<th>% Difference From Experimental</th>
</tr>
</thead>
<tbody>
<tr>
<td>TT0</td>
<td>Experimental Data FC.C M =0.4</td>
<td>0.2817</td>
<td>0%</td>
</tr>
<tr>
<td>TT1</td>
<td>Realizable k-ε ; Curvature Correction On</td>
<td>0.1943</td>
<td>36.69%</td>
</tr>
<tr>
<td>TT2</td>
<td>Realizable k-ε ; Curvature Correction Off</td>
<td>0.1951</td>
<td>36.33%</td>
</tr>
<tr>
<td>TT3</td>
<td>Realizable k-ε ; 2nd Order; Quadratic</td>
<td>0.2244</td>
<td>22.66%</td>
</tr>
<tr>
<td>TT4</td>
<td>Realizable k-ε ; 2nd Order; Cubic</td>
<td>0.2260</td>
<td>21.95%</td>
</tr>
<tr>
<td>TT5</td>
<td>k-ω SST; Curvature Correction On</td>
<td>0.2396</td>
<td>16.16%</td>
</tr>
<tr>
<td>TT6</td>
<td>k-ω SST; Curvature Correction Off</td>
<td>0.2398</td>
<td>16.07%</td>
</tr>
<tr>
<td>TT7</td>
<td>Elliptic Blend k-ε</td>
<td>0.2215</td>
<td>23.93%</td>
</tr>
</tbody>
</table>
Figure 4-14: Global Results Comparison for Each Turbulence Test for FC.C Geometry at \( M = 0.4 \)

Figure 4-15: All Turbulence Model Testing Against Experimental Data for FC.C \( M=0.4 \)
Figure 4-16: Far Wall Based Turbulence Models Tested Against Experimental Data for FC.C M=0.4
Figure 4-17: Near Wall Based Turbulence Models Tested Against Experimental Data for FC.C M=0.4
From the results gathered within the turbulence testing it was decided to use k-ω SST with the Curvature Correction option turned off for the remainder of this study. This model was chosen because it provided the best comparison against experimental data in regards to overall average effectiveness in a global and span wise view as well as slope comparison of effectiveness immediately after the film holes. In addition, as stated in the literature (Raj P.Y., 2013), this model has been used in similar geometry set ups as this current work and has provided the best results in regards of calculating adiabatic effectiveness. The reason k-ω SST is expected to perform the best in both global comparison as well as locally immediately downstream of the film holes, is because this turbulence model was designed to combine k-ε ‘s free stream accuracy far from the wall with the near wall accuracy of k-ω. Thus, k-ω SST was capable of best predicting the interaction of the film jets with the free stream as compared to the other turbulence models.
Also of note, the results depicted on Figure 4-18 highlight the capabilities of each turbulence model on their ability to capture the interaction of the flow close to the adiabatic wall film hole test section. From this figure, it displayed how turbulence models that were created for near wall accuracy, such as k-ω SST and EBK-ε, all did the best in regards to producing the slope of descending effectiveness immediately downstream of the film jets. This predication capability of the descending adiabatic effectiveness slopes is important because it displays how these turbulence models are capable of predicting the mixing and reattachment of the flow to the adiabatic test surface. However, Figure 4-18 also displayed how turbulence models that were made fundamentally to handle flows further from the wall, such as Realizable k-ε, were not able to predict the flow immediately following the film hole jets.

As for each turbulence models capabilities to produce the overall average magnitude of adiabatic effectiveness can be seen in Figure 4-14, Figure 4-15, Figure 4-16, and Figure 4-17, it was found that overall turbulence models that are fundamentally based to handle near wall flow mixing, such as k-ω SST and EBK-ε, were more accurate then far wall based turbulence models such as Realizable k-ε. However, as can be seen in Figure 4-14, there is an improvement in overall percent difference in the results of Realizable k-ε when additional turbulence parameters are included to help capture vorticity and rotation of the flow. Realizable k-ε with the added quadratic option is meant to improve the performance of RKE’s capability to capture secondary flows and vorticity. The addition of the cubic option for RKE is made to include a velocity gradient function that aids in the capability for capturing streamline curvature and rotation of the flow. Thus, since high mixing and shearing of the fluid is expected a near wall regions since film jet coolant is being ejected at an offset angle relative to the freestream, additional options to aid RKE’s capability to
analyze rotation and vorticity would improve this turbulence models ability to accurately represent the flow phenomenon. However, despite these improvements to the RKE turbulence model with the quadratic and cubic options, concerning magnitude prediction of adiabatic effectiveness, it still did not perform as well as the k-ω SST or EBK turbulence model which are turbulence models that are meant to capture flow characteristics near the wall. This improvement of near wall based turbulence models predicting magnitude of adiabatic effectiveness better then far wall based turbulence models is highlighted best in Figure 4-17. In this graph, the plots displayed in gray represent the far wall based turbulence models, while the colored plots display the near wall based turbulence models and their comparison to the experimental data.

In addition, as can be seen in both Figure 4-14 and Figure 4-18, the curvature correction setting did negligible change within this study for both RKE and k-ω SST. This turbulence setting was used in order to capture any major effects the compound angled jet flow relative to the free stream flow would cause in the turbulence models capability of predicting the effectiveness.

However, in order to highlight the capabilities of k-ω SST in producing the same results as found in the experimental data, a local comparison was done comparing the results produced from RKE and k-ω SST to the actual experimental data for the FC.C geometry at a blowing ratio of M = 0.4 which can be seen in Figure 4-19. RKE was used in this comparison since it was unable to capture both the overall magnitude and slope of the adiabatic effectiveness as compared to k-ω SST. By referencing the arrows on Figure 4-19, it can be seen that the overall local magnitude produced by k-ω SST resembles the experimental data much more accurately then RKE. This difference in magnitude accuracy that can be seen visually is the reason why the results produced by k-ω SST are within a smaller overall percent difference range then the RKE model which was
shown in Figure 4-14, Figure 4-15 and Figure 4-18. Figure 4-19 also displays the visual local effects of k-ω SST capability in replicating the local slope decline immediately after the film holes as shown in Figure 4-18. This phenomenon is captured by the overall length of the high effectiveness tail immediately trailing the film holes. As shown in the figure below, RKE does not replicate the overall length and magnitude of the coolant immediately trailing the film hole as seen in k-ω SST and the experimental data. This is why, when referring to Figure 4-18, the drop in average effectiveness trailing each film hole location was much steeper for the RKE model then what was shown in both the k-ω SST and the experimental data.

**Figure 4-19: Comparison of CFD Results to Experimental FC. C M=0.4**

Overall k-ω SST without the curvature correction option performed the best in regards to all comparisons tested within this study. k-ω SST predicted the overall global adiabatic effectiveness within 16.07% difference of the experimental as shown in Figure 4-14, the magnitude of the spanwise average adiabatic effectiveness within 0% difference at the first initial rows to
approximately 15% to the last rows as shown in Figure 4-17, and overall adiabatic effectiveness slope comparison immediately after film holes as shown in Figure 4-18. Thus, k-ω SST has proven itself to be a reliable turbulence model throughout these comparisons to replicate the experimental data which was described already to have a 10% uncertainty in their adiabatic effectiveness values gathered (Nathan & Dyson, 2014).

4.5 Geometry Comparison with Blowing Ratio

The current section aims to display k-ω SST’s capabilities to replicate the flow characteristics found in experiments done at the University of Central Florida (Natsui, 2012). The geometries and blowing ratios used to benchmark k-ω SST’s capabilities can be seen in Table 4-5 and their results compared to experimental data can be seen in Figures 4-20 through Figure 4-25.

As can be seen in Figures 4-20 and Figure 4-22, there was high accuracy in k-ω SST’s capabilities to replicate both the overall magnitude and local slope for adiabatic effectiveness for both the FC.A and FC.C geometry at a blowing ratio of M = 0.4. There was higher accuracy in replicating the FC.A geometry then the FC.C geometry at this blowing ratio since the FC.A geometry stayed within the 10% uncertainty range of the experimental data throughout the entire test section while the FC.C geometry increased in percent difference from nearly unity at the first few rows to approximately 15% at downstream row locations. The reason for k-ω SST’s better replication of the experimental data for the FC.A geometry then the FC.C geometry is because, as can be seen in Table 3-1, the inclination angle α is less for the FC.A geometry which helps prevent lift off of the coolant. This allows k-ω SST’s strength of calculating flow phenomenon near the wall to be fully utilized since less coolant will be farther away from the testing surface since the inclination angle α is smaller. This increase of coolant being injected farther away from the
adiabatic test section can be seen in Figures 4-27 and Figure 4-28 where it shows more cooling effects further away from the wall occur for the FC.C geometry then the FC.A geometry. This is an unwanted effect since any more coolant that enters the free stream cools down the ambient temperature which reduces turbine efficiency as well as taking away coolant effects that could have been used to cool the test section.

As blowing ratio increased for both the FC.A geometry and FC.C geometry, less accuracy was found concerning k-ω SST’s capability to replicate the downward slope of adiabatic effectiveness immediately after each film hole which can be seen in Figures 4-20 through Figure 4-24. This was concluded to be due to the fact that as blowing ratio increased, more and more coolant was ejected further and further into the freestream away from the adiabatic wall where k-ω SST struggled to predict how the flow will return back to the surface since this turbulence model is fundamentally based to work near the wall. A more detailed visual effect of this can be seen in the FC.C geometry for M = 0.8 on Figure 4-23 and Figure 4-29. Here the downward slope of the spanwise average adiabaticaic effectiveness immediately after the film holes is predicted well on the earlier rows where less coolant is reaching the free stream, but as more downstream rows are reached, the coolant begins to build up and more and more coolant is reaching further away from the test section which reduces the cooling effect captured immediately after the film holes.

Although the overall downward slope of the spanwise average adiabatic effectiveness prediction immediately after the film holes lacks with increasing blowing ratio, the overall magnitude prediction of the spanwise average adiabatic efficiveness becomes more consistent as blowing ratios increase for both geometries. For example, as can be seen in Figure 4-20 and Figure 4-22 for blowing ratio of M =0.4 for both geometries, the further downstream the flow location is,
the more the percent difference increases. However, for higher blowing ratios, as can be seen in Figure 4-21, Figure 4-23, and Figure 4-24, the overall difference in magnitude from the experimental data remains relatively constant. For a blowing ratio of $M = 0.8$, a correction factor that would increase the adiabatic effectiveness by 0.05 would shift the graph upwards enough to be entirely within the 10% uncertainty range of the experimental data. As for the blowing ratio of $M = 1.6$ cases, a correction factor to increase the adiabatic effectiveness by 0.08 would also shift the graph enough to be entirely within the 10% uncertainty range. This finding could be useful if the turbine blade designer wanted to have an overall understanding of the rate of change in adiabatic effectiveness for a set design of full coverage film cooling jets.

Although the overall magnitude was not predicted as well for higher blowing ratios than for lower blowing ratios (this could be accounted for by the offset correction factor as discussed above), k-ω SST was capable of displaying the presence of lift off at high blowing ratios ($M = 1.6$). As can be seen in Figures 4-21, Figure 4-24, Figure 4-35, and Figure 4-36 for the first few rows of film jets, there is minimal increase in adiabatic effectiveness for both geometries due to lift off. However downstream at approximately the 8th row of film holes, the coolant begins to return and reattach to the test surface which causes a fast rate in change in the adiabatic effectiveness which can be shown by the overall slope increase of the graph within this location on Figure 4-21 and Figure 4-24. However, this prediction in reattachment was found to be under predicted for locations further downstream. This can be shown when comparing Figure 4-25 and Figure 4-26 which show that for the experimental data, as blowing ratio increases, the average spanwise adiabatic effectiveness also increases downstream while for the CFD k-ω SST case of $M = 1.6$, this higher blowing ratio was under the average spanwise effectiveness produced by $M = 0.8$. 
downstream. This could be attributed to the excess lift off of coolant predicted by k-\(\omega\) SST for the M =1.6 case as opposed to the M = 0.8 case as shown in Figure 4-31 and Figure 4-29 respectively. This excessive lift-off predicted at this blowing ratio showed some of the reattachment limitations of the k-\(\omega\) SST turbulence model for high blowing ratios. The localized effects of this lift-off phenomenon captured by k-\(\omega\) SST can be seen in Figure 4-30, Figure 4-31, Figure 4-35, and Figure 4-36.

**Table 4-5: Geometry and Blowing Ratios Tested for CFD Analysis**

<table>
<thead>
<tr>
<th>Geometry</th>
<th>Blowing Ratios Tested</th>
</tr>
</thead>
<tbody>
<tr>
<td>FC.A</td>
<td>M = 0.4 ; M = 1.6</td>
</tr>
<tr>
<td>FC.C</td>
<td>M = 0.4 ; M = 0.8; M =1.6</td>
</tr>
</tbody>
</table>

*Figure 4-20: Comparison of Experimental to CFD Results; FC.A M = 0.4*
Figure 4-21: Comparison of Experimental to CFD Results; FC.A M = 1.6

Figure 4-22: Comparison of Experimental to CFD Results; FC.C M = 0.4
Figure 4-23: Comparison of Experimental to CFD Results; FC.C M = 0.8

Figure 4-24: Comparison of Experimental to CFD Results; FC.C M = 1.6
Figure 4-25: Comparison of Various CFD Results for FC.C at Increasing M

Figure 4-26: Comparison of Experimental Results for FC.C with Increasing Blowing Ratios M (Natsui, 2012)
Figure 4-27: $k$-ω SST Adiabatic Cooling Effectiveness Distribution for FC.A $M=0.4$

Figure 4-28: $k$-ω SST Adiabatic Cooling Effectiveness Distribution for FC.C $M=0.4$
Figure 4-29: k-ω SST Adiabatic Cooling Effectiveness Distribution for FC.C M=0.8

Figure 4-30: k-ω SST Adiabatic Cooling Effectiveness Distribution for FC.A M=1.6
Figure 4-31: k-ω SST Adiabatic Cooling Effectiveness Distribution for FC.C M=1.6

Figure 4-32: k-ω SST Adiabatic Cooling Effectiveness Distribution Locally for FC.A M=0.4

Figure 4-33: k-ω SST Adiabatic Cooling Effectiveness Distribution Locally for FC.C M=0.4

Figure 4-34: k-ω SST Adiabatic Cooling Effectiveness Distribution Locally for FC.C M=0.8
Figure 4-35: k-ω SST Adiabatic Cooling Effectiveness Distribution Locally for FC.A M=1.6

Figure 4-36: k-ω SST Adiabatic Cooling Effectiveness Distribution Locally for FC.C M=1.6
CHAPTER 5: CONCLUSION

Based off the results produced from this study, it can be concluded that the k-ω SST turbulence model can perform an accurate representation of capturing flow phenomenon for lower blowing ratios with minimal differences. However, for higher blowing ratios, the use of a correction factor to raise the magnitude in effectiveness locally (to aid in the lack of capture of coolant reattachment predicted) would provide a similar representation of what will be found experimentally.

In addition, it was also shown that the use of turbulence models that were created to capture near wall flow interactions are capable of predicting the phenomena for full coverage film cooling array, while turbulence models that were not meant specifically for near wall interaction could not accurately predict the interaction of the coolant flow near the wall.
REFERENCES


