Adiabatic Film Cooling Effectiveness of a Transpiration-Cooled Leading Edge Fabricated by Laser Additive Manufacturing

2018

Luisana Calderon

Find similar works at: https://stars.library.ucf.edu/etd

University of Central Florida Libraries http://library.ucf.edu

Part of the Mechanical Engineering Commons

STARS Citation


This Masters Thesis (Open Access) is brought to you for free and open access by STARS. It has been accepted for inclusion in Electronic Theses and Dissertations by an authorized administrator of STARS. For more information, please contact lee.dotson@ucf.edu.
ADIABATIC FILM COOLING EFFECTIVENESS OF A TRANSPIRATION-COOLED LEADING EDGE FABRICATED BY LASER ADDITIVE MANUFACTURING

by

LUISANA CALDERON
B.S.M.E., University of Central Florida, 2016

A thesis submitted in partial fulfilment of the requirements for the degree of Master of Science in Mechanical Engineering in the Department of Mechanical and Aerospace Engineering in the College of Engineering and Computer Science at the University of Central Florida Orlando, Florida

Fall Term
2018
ABSTRACT

Laser additive manufacturing (LAM) is an emerging technology capable of fabricating complex geometries not possibly made by investment casting methods for gas turbine applications. LAM techniques consist of building parts in a layer-by-layer process by selectively melting metal powders. In the present study, a mock leading edge segment of a turbine blade fabricated by LAM of Inconel 718 powders is investigated. For this particular design, the traditional showerhead film cooling holes have been replaced by two strips containing engineered-porous regions with the purpose of simulating the effect of transpiration cooling.

Transpiration cooling has been considered a promising external convective cooling method capable of providing a more uniform film and higher adiabatic film cooling effectiveness than conventional discrete film cooling. In addition, many studies have shown that this technique can yield high firing temperatures with much less coolant consumption than discrete film cooling. In this current study, adiabatic film cooling effectiveness is investigated by means of mass transfer using pressure sensitive paint (PSP). The experiments are conducted for blowing ratios ranging between $M = 0.03$ and $M = 0.28$ for a nominal density ratio of 1.5. The density ratio is obtained by using air as the mainstream flow and $CO_2$ as the secondary flow (or coolant source). Results indicate higher coverage and film cooling effectiveness when increasing blowing ratio at the expense of higher pressure drop. In addition, the experimental results are compared to numerical analyses performed using steady state Reynolds Average Navier Stokes (RANS) simulations.
ACKNOWLEDGMENTS

My special thanks Dr. Jayanta Kapat for his guidance and support through my graduate studies and for making this work possible. I thank the members of the Center for Advanced Turbo-machinery and Energy Research (CATER) specially Brandon Ealy, Andres Curbelo, Patrick Tran, and Husam Zawati for their friendship and support. I would also like to thank Dr. Erik Fernandez for being a good mentor in our lab.

I would like to acknowledge the IRES (International Research Experience for Students) program for extending this work thanks to the guidance of Dr. Seetha Raghavan from UCF and Dr. Marion Bartsch from the German Aerospace Center (DLR).

Lastly, I would also like to thank Dr. Ilya Mingareev and acknowledge the Fraunhofer Institute for Laser Technology for making the manufacturing of the test article possible, and North Star Imaging for conducting the non-destructive testing.
# TABLE OF CONTENTS

LIST OF FIGURES ................................................................. viii

LIST OF TABLES ................................................................. xi

NOMENCLATURE ................................................................. xii

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

1.1 Brayton Cycle ................................................................. 1

1.2 Turbine Cooling ............................................................ 3

   1.2.1 Internal Cooling ....................................................... 5

   1.2.2 External Cooling ...................................................... 5

      1.2.2.1 Film Cooling .................................................... 6

      1.2.2.2 Transpiration Cooling .......................................... 7

      1.2.2.3 Heat Transfer to a Filmed-Cooled Boundary layer ........... 10

      1.2.2.4 Independent Parameters used in Film Cooling ............... 11

1.3 Goals of the study ......................................................... 12

1.4 Conventional Manufacturing ........................................... 12

1.5 Laser Additive Manufacturing (LAM) .................................. 13

CHAPTER 2: TEST ARTICLE DESIGN AND NON-DESTRUCTIVE EVALUATION .. 16

2.1 Test Article Design ....................................................... 16

2.2 Test Article Manufacturing ............................................. 17

2.3 Non-Destructive Testing ............................................... 20

CHAPTER 3: METHODOLOGY .................................................. 23

3.1 Experimental Setup ...................................................... 23
CHAPTER 4: ADIABATIC FILM COOLING EFFECTIVENESS

4.1 The Mass and Heat Transfer Analogy ................................................. 28
4.2 Pressure Sensitive Paint ................................................................. 30
4.3 PSP Calibration ............................................................................. 31
4.3.1 PSP Calibration Setup ............................................................... 31
4.3.2 PSP Calibration Procedure ......................................................... 32
4.4 Effectiveness from Pressure Ratio ................................................... 33
4.5 PSP Processing .............................................................................. 33
4.6 Uncertainty Analysis ..................................................................... 34
4.7 Numerical Setup and Boundary Conditions ...................................... 34
4.8 Validation Case .............................................................................. 37

CHAPTER 5: RESULTS ................................................................. 40

5.1 Local Adiabatic Film Cooling Effectiveness Comparison of Results ........ 41
5.2 Lateral Averaged Effectiveness Comparison of Results ......................... 45
5.3 Discharge Coefficient ................................................................. 48

CHAPTER 6: FUTURE WORK ........................................................ 52

6.1 Manufacturing Considerations ....................................................... 52
6.2 DLR Testing Goals ................................................................. 55

CHAPTER 7: CONCLUSION ............................................................ 57

APPENDIX A: EXPERIMENTAL SETUP ........................................ 59
LIST OF FIGURES

Figure 1.1: Brayton Cycle T-S Diagram .................................................. 2
Figure 1.2: Effect of Pr in the Brayton Cycle; (a) Ideal Brayton cycle, (B) Pr vs. Specific Work ......................................................... 3
Figure 1.3: Temperature Profile of Blade Cross-Section [6]. ...................... 4
Figure 1.4: (a) Discrete Film Cooling; (b) Transpiration Cooling ................. 7
Figure 1.5: Combination of Film Cooling and Transpiration Cooling [24] .......... 9
Figure 1.6: Relative coolant-flow ratio vs. TIT [11] .................................. 10
Figure 1.7: Investment Casting of Turbine Blades/Vanes [1] .......................... 13
Figure 1.8: Ceramic core (left); Wax mold with inner core (middle); Final cast blade with internal cooling features (right) [22] ....................................... 14
Figure 1.9: LAM Process (top [32]) .......................................................... 15
Figure 2.1: Leading edge test article dimensions: (a) mock leading edge; (b) leading edge cross section; (c) internal engineered porous structure .................... 17
Figure 2.2: Preliminary build with supporting structures (left) and actual part (right) . 18
Figure 2.3: Test article showing outer holes (left) and impingement holes (right) .... 19
Figure 2.4: Test article showing the lattice (top) and internal impingement cavity (bottom) 19
Figure 2.5: Surface variance [7] ................................................................. 21
Figure 2.6: Channel variance of the impingement holes [7] ............................. 22
Figure 2.7: Concentricity of impingement holes in the spanwise direction for different θ locations [7] ............................................................. 22
Figure 3.1: Test Article and after-body attachment: (a) Top view and (b) Side view . 24
Figure 3.2: Film Cooling Wind Tunnel: (a) Isometric view and (b) Cross sectional view . 25
Figure 3.3: Acceleration parameter K ......................................................... 26
Figure 3.4: Wind tunnel side-wall pressure ........................................... 27

Figure 4.1: PSP calibration at different temperatures. ................................. 30
Figure 4.2: PSP calibration setup ............................................................. 32
Figure 4.3: Adiabatic film cooling effectiveness uncertainty tree .................... 35
Figure 4.4: Blowing ratio uncertainty tree .................................................. 36
Figure 4.5: Repeatability of effectiveness at various blowing ratios: \( M = 0.03, M = 0.12, \) and \( M = 0.28 \) .......................................................... 36
Figure 4.6: Computation domain and boundary conditions ......................... 37
Figure 4.7: Comparison of adiabatic film-cooling effectiveness with correlations found in literature for \( M = 0.03 \) .......................................................... 39

Figure 5.1: Plane A and plane B PSP location ........................................... 41
Figure 5.2: Lateral average effectiveness comparison between plane A and plane B 42
Figure 5.3: Experimental effectiveness distribution at various blowing ratios: (a) \( M = 0.03 \), (b) \( M = 0.06 \), (c) \( M = 0.12 \), (d) \( M = 0.28 \) ......................... 44
Figure 5.4: Numerical effectiveness distribution at various blowing ratios: (a) \( M = 0.03 \), (b) \( M = 0.06 \), (c) \( M = 0.12 \), (d) \( M = 0.28 \) ......................... 45
Figure 5.5: Numerical effectiveness distribution at \( y'/D = 0 \) for various blowing ratios: (a) \( M = 0.03 \), (b) \( M = 0.06 \), (c) \( M = 0.12 \) and (d) \( M = 0.28 \) ......................... 46
Figure 5.6: Lateral average effectiveness comparison between experiment and CFD for various blowing ratios .................................................. 47
Figure 5.7: Surface average effectiveness comparison .................................. 48
Figure 5.8: Discharge coefficient comparison .............................................. 50
Figure 5.9: Pressure drop comparison ....................................................... 51

Figure 6.1: LAM tensile specimen ............................................................. 53
Figure 6.2: Tensile specimen inner lattice design ................................. 53
Figure 6.3: Tensile specimen structure .................................................. 54
Figure 6.4: 3D-printed Trial Test .............................................................. 55
Figure 6.5: Schematic of the experimental setup ...................................... 56

Figure A.1: $\text{CO}_2$ Setup ................................................................. 60
Figure A.2: Experimental Setup ............................................................. 60

Figure B.1: Mesh cross-section scene ..................................................... 62
Figure B.2: Mesh scene for different cross-sections .................................. 62
Figure B.3: Cell surface cross-section scene .......................................... 63

Figure C.1: Numerical effectiveness distribution at $M = 0.03$ for various planar locations: (a) $y'/D = 0.04$ (b) $y'/D = 0$, (c) $y'/D = -0.04$ ...................... 65
Figure C.2: Numerical effectiveness distribution at $M = 0.06$ for various planar locations: (a) $y'/D = 0.04$ (b) $y'/D = 0$, (c) $y'/D = -0.04$ ...................... 65
Figure C.3: Numerical effectiveness distribution at $M = 0.12$ for various planar locations: (a) $y'/D = 0.04$ (b) $y'/D = 0$, (c) $y'/D = -0.04$ ...................... 66
Figure C.4: Numerical effectiveness distribution at $M = 0.28$ for various planar locations: (a) $y'/D = 0.04$ (b) $y'/D = 0$, (c) $y'/D = -0.04$ ...................... 66
LIST OF TABLES

Table 2.1: Test Article Manufacturing Specifications .................................. 18

Table 3.1: Experimental conditions ......................................................... 24

Table 4.1: Mesh sensitivity analysis ......................................................... 38

Table 5.1: Blowing ratio with two area definitions .................................... 40
NOMENCLATURE

\( \dot{m} \) \hspace{1cm} Mass flow rate \([kg/s]\)

\( A \) \hspace{1cm} Surface area of the porous slot \([m^2]\)

\( C \) \hspace{1cm} Concentration

\( C_D \) \hspace{1cm} Discharge coefficient

\( CFD \) \hspace{1cm} Computational Fluid Dynamics

\( CO_2 \) \hspace{1cm} Carbon Dioxide

\( D \) \hspace{1cm} Diameter of the leading edge \([m]\)

\( d \) \hspace{1cm} Diameter of the holes \([m]\)

\( DIC \) \hspace{1cm} Digital Image Correlation

\( DR \) \hspace{1cm} Density Ratio

\( h \) \hspace{1cm} Convective Heat Transfer Coefficient

\( I \) \hspace{1cm} Intensity

\( IGT \) \hspace{1cm} Industrial Gas Turbine

\( IR \) \hspace{1cm} Intensity Ratio

\( J \) \hspace{1cm} Momentum Flux Ratio

\( K \) \hspace{1cm} Acceleration Parameter
**LAM**  Laser Additive Manufacturing

**Le**  Lewis Number

**M**  Blowing Ratio

**MW**  Molecular weight ratio = $MW_{CO_2}/MW_{air}$

**O$_2$**  Oxygen

**P**  Pressure [Pa]

**PR**  Pressure Ratio from PSP

**Pr**  Compressor Pressure Ratio

**q''**  Convective Heat Flux

**Re**  Reynolds Number

**s**  Transpiration Slot Arclength [m]

**SLM**  Selective Lase Melting

**T**  Temperature

**TBC**  Thermal Barrier Coating

**TLC**  Thermochromic Liquid Crystal

**TR**  Temperature Ratio

**U**  Velocity [m/s]

**x**  Absolute Streamwise Direction [m]

**x, y, z**  Absolute Directions [m]
**XRD**  X-Ray Diffraction

\( y \)  Absolute Spanwise Direction [m]

\( y' \)  Relative Spanwise Direction [m]

\( \Delta P \)  Pressure drop [Pa]

**Greek Symbols**

\( \alpha \)  Thermal Diffusivity

\( \bar{\eta} \)  Surface Average Effectiveness

\( \beta \)  Calibration Coefficient

\( \delta \)  Mass Diffusivity

\( \eta_{Brayton,\text{ideal}} \)  Ideal Brayton Cycle Thermal Efficiency

\( \eta_{th} \)  Thermal Efficiency

\( \infty \)  Freestream

\( \nu \)  Kinematic Viscosity [m\(^2\)/s]

\( \rho \)  Density [kg/m\(^3\)]

\( \theta \)  Angle

\( \eta \)  Effectiveness

\( \gamma \)  Ratio of Specific Heats

**Subscripts**

\( a \)  Absolute Pressure
allowable  Allowable Temperature

avg  Average

aw  Adiabatic Wall

bg  Background

c  Coolant

d  Diameter of the holes

FC  Film Cooling

fg  Foreign gas (CO₂)

mix  Fluid mixture (air - CO₂)

o  Without Film Cooling

ref  Reference

T  Transpiration

t  Total

w  Wall
CHAPTER 1: INTRODUCTION

According to the International Energy Outlook 2017, renewables are the fastest growing energy source projected through 2040. Although the demand for coal is expected to reduce by that year, fossil fuels are still projected to contribute with the majority of the world’s energy consumption with natural gas listed as the fastest-growing fossil fuel. As renewables are the future, we need to be able to supply very large power consumption demands. In order to do that, it is necessary to utilize gas turbines as they can be quickly ramped up and down to meet high peak demands of power paving the way for renewable energy during off-peak demand hours.

1.1 Brayton Cycle

Vasts amounts of research have been conducted through the years to increase thermal cycle efficiencies of gas turbines by extracting the maximum amount of work per unit output. This is highly beneficial for the industry as well as consumers, as a very small percentage increase in the thermal efficiency can help to bring down the fuel cost over time, therefore lowering the price of electricity in addition to offering positive environmental factors.

Gas turbines are a type of heat engine whose cycle efficiency is found from the Carnot efficiency, (1.1)

\[ \eta_{th} = 1 - \frac{T_1}{T_3} \]  

(1.1)

The Carnot efficiency is the theoretical maximum efficiency that can be achieved by a heat engine which will never reach unity; this efficiency can be improved by increasing the ratio of \( \frac{T_1}{T_3} \).

Gas turbines are governed by the thermodynamic Brayton cycle, which can be represented by a T-S diagram (see Figure 1.1). For an ideal reversible process, intake air is adiabatically compressed through the compressor (1 → 2); then heat is added at a constant pressure (2 → 3);
combusted gases expand adiabatically through the turbine ($3 \rightarrow 4$); and for a closed-loop system, the gases release energy at constant pressure where the cycle repeats ($4 \rightarrow 1$).

\[
\eta_{Brayton,ideal} = 1 - \frac{1}{Pr^{\gamma-1}}
\]  

(1.2)

The ideal Brayton cycle efficiency is derived in terms of the engine’s pressure ratio, $Pr$, as seen in Equation (1.2). Lets now consider two cases where both temperatures are maintained at the same maximum temperature $T_3$ (Figure 1.2). Cycle A provides the highest thermal efficiency due to the increase in pressure ratio ($1 \rightarrow 2'$), while Cycle B has a lower thermal efficiency but
provides higher specific power output. Therefore, there is an optimal $Pr$ typically used in gas turbines due to its impact in the specific work. In consequence it is beneficial to increase the turbine inlet temperature (TIT) as much as possible for high thermal cycle efficiencies and specific power output.

Figure 1.2: Effect of Pr in the Brayton Cycle; (a) Ideal Brayton cycle, (B) Pr vs. Specific Work

1.2 Turbine Cooling

The TIT in modern gas turbines for power generation are surpassing the material’s limitations making it challenging for thermal engineers to keep the most critical components in the turbine under safe operating conditions. The goal is to keep increasing the TIT (also known as the allowable temperature ($T_{allowable}$) or firing temperature) for higher thermal efficiencies. Therefore, there is a need for protective thermal barrier coatings (TBC’s) as well as cooling technologies to protect the parts in the hot-gas path. IGT’s are currently being operated up to 1600°C while the material’s limitations is much lower than that.

The thermal management of airfoils or other critical parts in a turbine is subdivided into
three main categories: that is materials, thermal barrier coatings (TBC’s), and cooling. To understand how important are the last two, let’s take a look at Figure 1.3 which displays the temperature profile across a turbine blade. Here, several layers are displayed including the base material made from an Inconel superalloy, the bond coat, and the TBC.

As the goal is to minimize the thermal resistance from the inner walls of the turbine blade, internal convective cooling schemes are often utilized for the first stages of turbine rotors and stators. On the other hand, introduction of thermal resistance is necessary in order to reduce the heat transfer from the hot gases to the external surfaces of these components allowing higher TIT’s. Therefore, the surfaces are coated with TBC’s due to its low thermal conductivity. In addition, external convective cooling methods are used to reduce the heat transfer even further by the use of
film cooling.

The cooling air utilized for both internal and external methods is bled from the last stages of the compressor reducing the overall available work. Therefore, one of the main goals of researchers is to maximize the efficiency of the cooling schemes as this directly impacts the power output of the turbine.

1.2.1 Internal Cooling

The heat removal process of blades and vanes occurs by the implementation of internal convective cooling methods applied within the internal walls. Heat transfer enhancement can be achieved by increasing the effective area or increasing the heat transfer coefficient. Typically, turbulators such as ribs, wedges, dimples/pimples and pin-fins are used in order to provide heat transfer enhancement. Turning passages with turbulators are typically placed in the mid-section of the blade, while pin-fins are used towards the trailing edge. One of the most critical sections of the blade is the leading edge, which is directly impacted by the combusted gases. A highly effective heat removal technique called impingement cooling is used to provide localized heat transfer enhancement. Impingement cooling consists of having a jet or series of jets of fluid flow hitting the critical surface therefore maximizing the heat transfer.

1.2.2 External Cooling

External convective heat transfer techniques such as film cooling has been widely used in turbine components such as blades, vanes, shrouds/platforms, etc., to carry the heat away from the hot surfaces. The principle is to reduce the heat transfer hence providing a thin insulating layer of cooler air on the target surface.
1.2.2.1 Film Cooling

Film cooling technologies have been implemented in IGT’s to extend the life of the most critical parts. In this concept, a jet in a cross-flow scenario is introduced hence a large research field has been dedicated for decades to study this complex interaction. Relative cooler fluid flow is injected through a hole or series of holes creating a coolant jet against the hot flow experienced by the surface (refer to Figure 1.4a). In the case of blades and vanes, film cooling holes are applied in a discrete manner to increase the film cooling effectiveness at the leading edge, tip and/or at the endwall. The effectiveness of this technique will be explained through this work, but for now, let us have a physical interpretation. Eckert and Esger [9] discuss how flow mixing occurs downstream the injection region consequently decreasing the film cooling effectiveness. At a certain distance from the injection point, more cooling holes can be added in order to keep replenishing the coolant flow hence providing higher film coverage. Patterns of film cooling holes is what is referred to as multi-row film cooling. Multi-row film cooling is also implemented in combustor liners due to the high heat loads experienced by these components. Apart from the shear layer mixing which leads to a rapid film effectiveness decay, jet lift-off is one of the drawbacks of discrete film cooling as the jet has the tendency of lifting up at higher coolant flow rates. The area under the lift-off region is of major concern due to its lack of effectiveness. Therefore, major research has been done to understand this phenomenon and how this can be avoided by changing and optimizing the geometrical parameters as well as flow conditions. In addition, the high thermal gradients associated with having uncooled areas or spots in between the holes can affect the mechanical properties of the part. Ealy at al. [6] explains how the life of the component can be reduced due to the presence of thermal gradients leading to creep deformation.
1.2.2.2 Transpiration Cooling

Transpiration cooling is a variation to film cooling where coolant fluid travels through a porous media (Figure 1.4b). Coolant flow is bled through a porous or “breathing” wall where fluid settles onto the hot surface therefore providing a protective cooling layer. The porous wall serves to induce pressure loses reducing the flow momentum at which the coolant leaves the surface; as a consequence fluid would tend to stay attached even at a high coolant flow rates providing an “ideal” attached film. As opposed to discrete film cooling where flow mixing reduces the effectiveness downstream injection, coolant fluid is continuously renewed for transpiration-cooled walls, therefore providing highly uniform temperature distributions [9].

Investigations on transpiration cooling date back from the 50’s. Goldstein et al. [14] experimentally investigated the adiabatic film cooling effectiveness of a transpiration-cooled segment over a flat plate and compared it to other studies. Eckert and Linvinwood [10] performed numerical investigations comparing transpiration and film cooling. They describe the superiority of transpiration cooling for both laminar and turbulent flow. Through analytical calculations, they also found lower heat transfer coefficients from transpiration cooling in comparison to other alternatives. They explain the negative effects radiation has on transpiration cooling, however it is
mentioned how this can be neglected in GT’s. Colladay [5] describes transpiration cooling as one of the most efficient cooling methods due to its high convective effectiveness and film uniformity. Esgar and Eckert [9] consider transpiration cooling to be more effective than film cooling and discuss the reduction in heat transfer coefficient occurring from the thickening of the transpiration-cooled boundary layer at higher TIT’s. Other transpiration cooling and full coverage film cooling studies are experimentally studied by Eckert and Cho [8].

Aerodynamic performance is directly affected by having a thicker boundary layer experienced by transpiration-cooled geometries in comparison to conventional film-cooled parts. Natsui et al. [24] conducted experiments coupling both cylindrical discrete film cooling holes with a segment of porous/permeable wall to find the most optimal case in terms of both heat transfer and aerodynamic efficiency (Figure 1.5). Despite the increase in aerodynamic losses when combining both cooling methods, an increase in downstream and lateral effectiveness was observed with a small percentage in additional cooling. This study founds the most promising design to be the downstream placement of the porous region at low blowing ratios due to having the highest surface average effectiveness. Other experimental studies performed by Natsui et al. [26] found that the combination of the two technologies offers a more efficient use of coolant. The lower coolant requirement of transpiration cooling in comparison to film cooling was also observed in early studies [5, 10, 11] (refer to Figure 1.6).

Although transpiration has shown to outperform traditional film cooling, manufacturing limitations must be taken into account, as porous structures cannot be manufactured using conventional methods such as investment casting. The application of transpiration cooling in gas turbine components has not been implemented due to the low strength obtained from sintered materials and the inability to control the porous structures [17]. The recent development of additive manufacture (AM) methods has opened the door for a new research field. Through these unconventional techniques, it is possible to fabricate very complex geometries such as porous structures. The following sections will cover the process for the fabrication of turbine blades/vanes; in addition, as an
alternative to investment casting, an additive manufacturing technique will be discussed with the objective of fabricating transpiration-cooled surfaces.

Figure 1.5: Combination of Film Cooling and Transpiration Cooling [24]
1.2.2.3 Heat Transfer to a Filmed-Cooled Boundary layer

Newton’s Law of Cooling is used to determine the convective heat flux. For flow over a surface with no film cooling, Equation 1.3 can be implemented using the freestream temperature, $T_\infty$, and the surface temperature, $T_{wall}$.

$$q''_0 = h_0(T_\infty - T_{wall}) \quad (1.3)$$

For a film-cooled wall, the heat flux is found using the temperature of the film, $T_f$, as seen in Equation 1.4.

$$q''_f = h_f(T_f - T_{wall}) \quad (1.4)$$

It is assumed that for an adiabatic wall ($q''_f = 0$), $T_f$ is equal to the adiabatic wall tempera-
ture, $T_{aw}$. A very important parameter of interest in film cooling performance is the adiabatic film cooling effectiveness $\eta$, which is utilized to normalize $T_{aw}$ (refer to Equation 1.5). For $\eta = 0$ the film temperature reaches the freestream temperature providing no effectiveness; similarly, for $\eta = 1$, the film temperature reaches that of the film coolant ejection temperature, $T_c$.

$$
\eta = \frac{T_\infty - T_{aw}}{T_\infty - T_c}
$$  \hspace{1cm} (1.5)

1.2.2.4 Independent Parameters used in Film Cooling

One of the most important independent parameters to discuss is the density ratio, $DR$, between the secondary fluid (coolant) and the cross-flow (freestream) as defined in Equation 1.6. Density ratio values representative of IGT’s range between $\sim 1.5$ and $\sim 2$ with its inversely propotional temperature ratios ranging between $\sim 0.5$ and $\sim 0.85$ [15]. Some experiments are conducted with $DR$ values lower than unity due to experimental setup limitations. In other cases, it is possible to achieve higher density ratios when injecting a foreign fluid as the coolant.

$$
DR = \frac{\rho_c}{\rho_\infty} \approx \frac{1}{TR}
$$  \hspace{1cm} (1.6)

$$
TR = \frac{T_c}{T_\infty}
$$  \hspace{1cm} (1.7)

Another parameter considered in film cooling performance is the blowing ratio or also called mass flux ratio, $M$, found in Equation 1.8. The blowing ratio defines the amount of mass flux of the coolant to the mass flux of the freestream accounting for both densities and velocities. Both blowing ratio and density ratio define the momentum flux ratio, $J$, as shown in Equation 1.9. This parameter describes the momentum flux of the coolant to that of the freestream. For high values of momentum flux ratios, the film cooling jet would have the tendency of protruding into the core therefore lifting off from the surface. As discussed earlier, jet lift-off is undesired as this
leaves areas of lower performance hence lower effectiveness.

\[ M = \frac{(\rho U)_c}{(\rho U)_\infty} = \frac{(\dot{m}/A)_c}{(\rho U)_\infty} \quad (1.8) \]

\[ J = \frac{(\rho U^2)_c}{(\rho U^2)_\infty} = \frac{M^2}{D_R} \quad (1.9) \]

1.3 Goals of the study

The demand for highly complex and efficient cooling geometries has increased the attention for additive manufacturing technologies. This is because many of these well-known cooling features can’t be manufactured using investing casting. Therefore, GT companies are investing in additive manufacturing techniques to enable certain cooling geometries that have shown to outperform those that are conventional. The focus on this work is to use laser additive manufacturing (LAM) to enable transpiration cooling as it has shown to perform better than traditional discrete film cooling.

The goal of this study is to investigate the adiabatic film cooling effectiveness of a transpiration-cooled mock leading edge (LE) segment of a turbine blade fabricated from Inconel 718 powders. The experiments are conducted using PSP (Pressure Sensitive Paint) using $CO_2$ as the coolant to achieve a nominal density ratio of 1.5.

1.4 Conventional Manufacturing

Investment casting is the conventional manufacturing method of choice for the fabrication of turbine blades and vanes. Several steps happen during the manufacturing process that can be costly and time consuming (refer to Figure 1.7). The process first starts by injecting wax into a master mold containing the desired part shape. For internally-cooled parts, a ceramic core containing the cooling features is first introduced before the wax injection process (see Figure 1.8). The
next step is to create an assembly of several of these parts before the sand-coating process to create the negative mold of the part followed by dewaxing. The assembly is often performed for aero turbine blades as these are small, however, for IGT blades, these step is done individually. Molten Inconel superalloy metal is poured into the shell created by the ceramic slurry. After the part is cast, the ceramic core is removed with chemicals while the shell is broken away.

Figure 1.7: Investment Casting of Turbine Blades/Vanes [1]

1.5 Laser Additive Manufacturing (LAM)

Laser additive manufacturing (LAM) or also known as selective laser melting (SLM) is an additive manufacturing technique that selectively melts metal powders with a laser beam to build parts in a layer-by-layer process. This process is very simple, and it first starts by having a platform bed with metal powders, through which a laser beam will be used to melt specified sections creating 2-dimensional slices of the desired part. After each layer is built, the platform
containing the powder bed will lower a specific amount in the order of micrometers, and the system will re-coat the bed with a new layer of powder; the process continues until a 3-dimensional volume reconstruction of the part is achieved (see process in Figure 1.9).

Some of the advantages of this process are as follows:

- Rapid prototyping
- Quick repair and optimization of parts
- Weight reduction
- Cost reduction of conventional methods
- Ability to fabricate complex geometries

Figure 1.8: Ceramic core (left); Wax mold with inner core (middle); Final cast blade with internal cooling features (right) [22]
Figure 1.9: LAM Process (top [32])
CHAPTER 2: TEST ARTICLE DESIGN AND NON-DESTRUCTIVE EVALUATION

2.1 Test Article Design

The present geometry, designed at the University of Central Florida (UCF), is developed to represent the leading edge section of a turbine blade (refer to Figure 2.1). The diameter of the part is $50\, mm$ with a span of $100\, mm$ forming a semi-cylindrical shape. An internal impingement sleeve containing 50 holes in the hub-to-tip direction and 25 holes in the $\theta$ direction is used to simulate internal impingement cooling. Apart for the internal array, a engineered-porous lattice structure is designed with the purpose of simulating transpiration cooling. These lattice structures are located within empty cavities at approximately $\pm 45\, \text{deg}$ from the stagnation region of the leading edge. Upstream and downstream of the cavities, there are arrays of holes where the fluid comes in and out, respectively. All the holes, including the lattice holes have a diameter, $d$, of $0.5\, mm$. The designed porosity of the cavities is found to be 0.57 obtained from dividing the void volume by the total volume. The lattice pattern consists of having 2 rows of cylinders arranged in the radial direction and 5 rows in the $\theta$-direction. The lattice pattern is organized in an inline configuration having a pitch of $6d$, while the exit holes extend $0.3d$ apart from each other in a staggered arrangement.
Figure 2.1: Leading edge test article dimensions: (a) mock leading edge; (b) leading edge cross section; (c) internal engineered porous structure

2.2 Test Article Manufacturing

One of the fabrication aspects of SLM parts that must be considered are supporting frames; they are required at the platform level from where the part is constructed as well as other locations if needed; these depend on the geometry itself as build angles above $\sim 60$ deg would require them, therefore compromising the porosity of the melt pool or the tolerance of the first layers; in addition, these can be utilized as means for heat dissipation during the fabrication process [6].

Different positions for the construction of the current test article were considered as build directions can affect the surface roughness of the part as well as the build quality. The final iteration was obtained by building the part 180 deg from the original position, therefore placing the supporting structures only inside the inner cavity as well as below the impingement sleeve (see Figure 2.2). The part was removed from the substrate by wire EDM (Electrical Discharge Machining), and removal of the remaining supports were done by unknown post-machining methods.
The NDE testing would serve us to better understand the impact supporting structures have on the overall surface quality.

The test article was manufactured at the Fraunhofer Institute of Laser Technology in Aachen, Germany. It was built using an EOS M270 SLM machine using Inconel 718 powders with a grain size diameter ranging between 25 – 45\(\mu m\). For other specifications of laser beam diameter and layer thickness refer to Table 2.1 (see images of the test article in Figures 2.3 and 2.4).

![Figure 2.2: Preliminary build with supporting structures (left) and actual part (right)](image_url)
Figure 2.3: Test article showing outer holes (left) and impingement holes (right)

Figure 2.4: Test article showing the lattice (top) and internal impingement cavity (bottom)
2.3 Non-Destructive Testing

The NDE evaluation was done via CT (Computer Tomography) X-rays by traversing it along the span of the part. This non-intrusive testing was conducted by North Star Imaging (NSI) using the state-of-the art NSI X5000 CT-Xray providing an image resolution of 100\(\mu m\). The VGStudio Max commercial software was used for the volume rendering; this software utilizes a fitting algorithm allowing the overlap between the 3D-volume reconstruction with the actual CAD to evaluate the differences or variance.

As previously mentioned, the design diameter of the holes and the intersecting cylinders of the engineered-porous lattice is only 0.5\(mm\). However, the actual dimension can vary up to \(\pm 0.12\)\(mm\) depending on the build direction and location of the holes. It is crucial to point out that there are major differences between the designed CAD and the actual part due to the manufacturing process. Positive variance is observed in the areas in contact with the supporting material having relative variance magnitudes of up to \(\pm 1d\) (Figures 2.5). Channel variance was also obtained through different \(z\)–planes for both the impingement array and porous structures (Figure 2.6).

It is noticeable the variance dependency on build angle as higher variance is observed as \(\theta \to 0\) deg, where \(\theta \approx 90\) deg is in the vertical direction of gravity. Ealy [7] also evaluated hole concentricity from the impingement array through various \(z\) and \(\theta\) locations (Figure 2.7). Similar observations are drawn as concentricity decreases as \(\theta \to 0\) deg and changes through different \(z\)–directions. It was therefore concluded that the best quality holes were observed at \(\theta \approx 90\) deg. These results agree very well with studies performed by Snyder et al. [31], where they investigated the relationship between build direction and build quality from microchannel coupons fabricated from DLMS (Direct Laser Metal Sintering). Similar to the results obtained by Ealy [7], the best quality microchannels were observed at the the vertical location with respect to gravity or \(\theta \approx 90\) deg.

It is expected to have a significantly much higher pressure drop from the numerical model
in comparison to experimental testing due to the actual part having higher effective roughness encompassing leftover residual material from supporting frames, and anomalies or imperfections associated with the machining process or material itself. These defects and imperfections can cause an impact in flow behavior as it is seen from the low discharge coefficients obtained from their study. Similarly to the behavior of the impinging array, it is expected for the showerhead film cooling holes to have poor concentricity as well as high distortion.

Figure 2.5: Surface variance [7]
Figure 2.6: Channel variance of the impingement holes [7]

Figure 2.7: Concentricity of impingement holes in the spanwise direction for different $\theta$ locations [7]
CHAPTER 3: METHODOLOGY

3.1 Experimental Setup

The downstream effectiveness was measured from a flat after-body enclosure added as an attachment to the end of the leading edge piece (see Figure 3.1). This enclosure, made from plexiglass acrylic, is also utilized to guide the coolant flow into the impingement sleeve section through a screen diffuser located 5.5\(D\) downstream the leading edge test article. This is to ensure an uniform coolant flow distribution before entering the impingement holes. In addition, all walls are capped and sealed in order to prevent coolant leakage. The entire test section is mounted on a 352.4\(mm\) x 429.4\(mm\) x 25.4\(mm\) plexiglass acrylic plate located at the bottom wall of the wind tunnel test section. The effectiveness measurements are collected in two different areas or planes away from both top and bottom walls of the test article in order to avoid edging effects. In late chapters, it will be discussed the placement of these planes and their relationship to film homogeneity.

3.1.1 Wind Tunnel

The mainstream air flow is supplied by a 15\(kW\) blower in an open-loop wind tunnel. The air flow is conditioned through a series of screens and a honeycomb as well as a 2D contraction. To reduce the boundary layer thickness at the test section a 2.2\(kW\) suction fan is used to bleed the freestream air before entering the test section. The wind tunnel is instrumented with three pitot probes, thermocouples and static pressure ports located at the side walls in order to conduct main flow velocity measurements. Pressure measurements are taken using an Omega HHP242 manometer and temperatures inside the wind tunnel are taken with a Fluke 51 II thermometer. The PSP images were taken by a CCD PCO camera of 1600 x 1200 resolution placed perpendicular to the test section along with an LED light to excite the paint molecules (refer to Figure 3.2).
Table 3.1: Experimental conditions

<table>
<thead>
<tr>
<th>Freestream Flow</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Re_{fs}</td>
<td>500,000</td>
</tr>
<tr>
<td>Ma_{fs}</td>
<td>0.1</td>
</tr>
<tr>
<td>Turbulence Intensity</td>
<td>&lt;0.010</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Coolant Flow</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>M</td>
<td>0.03 0.06 0.10 0.12 0.28</td>
</tr>
<tr>
<td>Re_{d}</td>
<td>300 710 1280 1540 3330</td>
</tr>
</tbody>
</table>

3.1.1.1 Pressure Gradients

The experiment is idealized to have zero-pressure gradients; however, there is a favorable pressure gradient occurring in the stream-wise direction allowing the flow to accelerate due to the
constant cross-sectional area in the test section from the boundary layer growth. To account for this, the local acceleration parameter $K$ is calculated according to Equation 3.1.

$$K = \frac{\nu}{(U_\infty)^2} \frac{dU_\infty}{dx}$$  \hspace{1cm} (3.1)
By assuming no total pressure loss in the core flow, the freestream velocity $U_\infty$ is found by the static pressure ports located at the side wall of the wind tunnel in the direction of the flow and by taking the total pressure from a pitot probes located upstream the test section. An acceleration parameter $K$ of $3.77 \times 10^{-10}$ is found (refer to Figure 3.3). Similarly, the static pressures are found along the wind tunnel test section displaying three sets of repeatability measurements (see Figure 3.4)

![Figure 3.3: Acceleration parameter K](image)

Figure 3.3: Acceleration parameter K
3.1.2 Coolant Supply

The secondary flow was supplied to the test article using CO$_2$ to provide a nominal density ratio of 1.5. The carbon dioxide is supplied as a cryogenic liquid from a 3500lb capacity Microbulk tank. A 30$kW$ electric vaporizer is utilized to change the liquid CO$_2$ to the gaseous state. The temperature is brought to the ambient temperature by passing the coolant through a second vaporizer (see A.1). An Omega FMA-1613A mass flow meter is used to measure the incoming coolant flow rate before entering the test section. A summary of the flow testing conditions are shown in Table 3.1.
CHAPTER 4: ADIABATIC FILM COOLING EFFECTIVENESS

This chapter introduces the theory behind the PSP technique by means of the mass and heat transfer analogy and its relationship to film cooling effectiveness. The PSP technique is explained in detail as well as the calibration setup and procedure. The uncertainty analysis of the adiabatic film cooling effectiveness and the blowing ratio are presented. In addition, the setup of the numerical model and boundary conditions are given along with a mesh sensitivity analysis. Lastly, towards the end of this chapter, a validation case study using the experimental and numerical results for the lowest blowing ratio $M = 0.03$ is obtained and compared against other experimental studies performed by Goldstein et al. [14] and Scesa et al. [30].

4.1 The Mass and Heat Transfer Analogy

The mass and heat transfer analogy has been widely used to calculate the adiabatic film cooling effectiveness since heat transfer effects are completely avoided. Different density ratios between the cross-flow and the secondary fluid can be obtained by introducing a foreign gas. Many experimental studies have been conducted through the years by methods of gas sampling. Pedersen et al. [28], Kacker et al. [18], Le Brocq et al. [19], and Salcudean et al. [29] used gas sampling to draw near wall mixtures by injecting a foreign gas into the cross-flow. Wright et al. [33] performed film cooling experiments using different techniques such as steady state PSP, TLC (Thermochromic Liquid Crystals), and TSP (Temperature Sensitive Paint). According to these studies, PSP showed superiority over the other methods as it was able to predict jet lift off by providing highly accurate results near the holes. Other experimental studies using PSP are found by Charbonnier et al. [4], Ahn et al. [2], Narzary et al. [23], Zhang et al. [34], and Shiou-Jiuan Li et al. [20].

The simplified two-dimensional governing equations for heat transfer and mass transfer in
a turbulent flow are found in Equations 4.1 and 4.2, respectively [16].

\[ G_x \frac{\partial T}{\partial x} + G_y \frac{\partial T}{\partial y} = \rho (\varepsilon_T + \alpha) \frac{\partial^2 T}{\partial y^2} \]  

\[ (4.1) \]

\[ G_x \frac{\partial C}{\partial x} + G_y \frac{\partial C}{\partial y} = \rho (\varepsilon_M + \delta) \frac{\partial^2 C}{\partial y^2} \]  

\[ (4.2) \]

\( \varepsilon_T \) - turbulent thermal diffusivity

\( \varepsilon_M \) - turbulent mass diffusivity

If the Lewis number is approximately 1, \( Le = \frac{\varepsilon_T + \alpha}{\varepsilon_M + \delta} \approx 1 \), then both temperature and species should have the same behavior [25]. This is the case for turbulent flows [16]. Therefore, the adiabatic film cooling effectiveness can be written as a ratio of concentrations of oxygen as displayed in Equation 4.3.

\[ \eta = \frac{T_\infty - T_{aw}}{T_\infty - T_c} = \frac{C_{O_2\infty} - C_{O_2w}}{C_{O_2\infty} - C_{O_2c}} = \frac{C_{O_2w}}{C_{O_2\infty}} \]  

\[ (4.3) \]

\( \varepsilon_T \) - turbulent thermal diffusivity

\( \varepsilon_M \) - turbulent mass diffusivity

In addition, having no concentrations of the gas tracer (or foreing gas) in the freestream \( (C_{fg\infty} = 0\%) \), the adiabatic film cooling effectiveness can also be defined as the concentration of the foreing gas relative to the total concentration injected \( (C_{fgc} = 100\%) \) [13]. Therefore, Equation 4.3 becomes Equation 4.4, and this is the definition used for the numerical simulations.

\[ \eta = \frac{C_{fg\infty} - C_{fgw}}{C_{fg\infty} - C_{fgc}} = \frac{C_{fgw}}{C_{fgc}} \]  

\[ (4.4) \]
4.2 Pressure Sensitive Paint

Pressure sensitive paint is an experimental technique used to measure the partial pressure of oxygen ($P_{O_2}$) on the surface. The paint is composed of a permeable binder containing luminescence molecules that emit an intensity value according to the amount of oxygen ($O_2$) molecules present in the binder. The excited state of the molecules can return to their ground state by oxygen quenching; therefore, the higher the air pressure above the permeable binder, the higher the $O_2$ concentration levels that quench or reduce the luminescence emissions [21].

![Figure 4.1: PSP calibration at different temperatures.](image)

As the intensity levels emitted from the PSP are a function of the air pressure (or $O_2$ concentration), it is important to define non-dimensional parameters to take into account reference states; these parameters are the intensity ratio ($IR$) and pressure ratio ($PR$) defined in Equations
4.5 and 4.6, respectively.

\[ IR = \frac{I_{\text{ref}} - I_{\text{bg}}}{I_{\text{test}} - I_{\text{bg}}} \]  \hspace{1cm} (4.5)

\[ PR = \frac{P}{P_{\text{ref}}} \]  \hspace{1cm} (4.6)

Once a relationship between \( IR \) and \( PR \) is obtained from the PSP calibration (Figure 4.1) the data points are fitted to a 3\textsuperscript{rd} order polynomial as suggested by Liu and Sullivan [21],

\[ PR = \beta_3 IR^3 + \beta_2 IR^2 + \beta_1 IR + \beta_0 \]  \hspace{1cm} (4.7)

Where, \( \beta_3, \beta_2, \beta_1, \) and \( \beta_0 \) are the coefficients found from the calibration.

4.3 PSP Calibration

The PSP is calibrated in-house at different temperatures and pressures. An aluminum coupon of 40\textit{mm} x 40\textit{mm} is painted with UniFIB PSP from Innovative Scientific Solutions Incorporated (ISSI). Following ISSI’s PSP application instructions, the coupon was painted using a Paashe TG-3F airbrush. Several coats were applied evenly through the coupon and the same was turned until a uniform finish was obtained. To reduce the temperature sensitivity of the PSP, the coupon was cured by heating it through its glass transition temperature of 60\textdegree C for 3 hours.

4.3.1 PSP Calibration Setup

The calibration was conducted inside of a calibration chamber to vary both the temperature and pressure of the PSP-painted coupon (refer to Figure 4.2). The coupon was instrumented with a thermoelectric Peltier heater and a thermocouple embedded into to aluminum at a point very close to the PSP surface. The calibration chamber was also instrumented with a thermocouple and two
pressure ports in order to monitor both temperature and pressure inside the chamber, respectively. A 0.4kW vacuum pump is used to allow calibrating at a range of various pressures. The coupon is completely visible from the outside of the chamber through a plexiglass acrylic window for full optical access. The PSP was excited using a 460nm LED light from ISSI and the images were captured by a 14-bit CCD PCO camera of 1600 x 1200 pixel resolution. To capture the correct wavelength emission from the PSP, a high pass filter is utilized.

![Figure 4.2: PSP calibration setup](image)

### 4.3.2 PSP Calibration Procedure

For any PSP experiment including its calibration, it is necessary to have background, reference and test intensity images. The background intensity ($I_{bg}$) images are taken to subtract any lighting present in the environment during the test. These background intensity images are first taken at ambient pressure having the wind-off condition (no flow) and the LED light off (equivalent to $\eta = 0$). The reference intensity images ($I_{ref}$) are recorded still with the wind-off condition
and the LED light on. Finally, the test intensity images \(I_{test}\) are taken with the wind-on condition and the LED light on by gradually lowering the pressure inside the chamber until the lowest possible pressure is obtained. For each run consisting of maintaining the coupon at a single temperature, a total of 20 images per set were recorded and averaged. The temperatures used for this calibration range between 12.2°C and 26.6°C in order to cover all testing conditions.

4.4 Effectiveness from Pressure Ratio

Carbon dioxide \((CO_2)\) is the foreign gas used as the coolant to achieve a density ratio of \(\sim 1.5\). Due to its absence of free \(O_2\) molecules, \(CO_2\) serves as a gas tracer, therefore allowing to capture the \(O_2\) molecules present at the target surface. Charbonnier et al. [4] explains in detail how to obtain the adiabatic film cooling effectiveness in terms of \(PR\), the value that can be obtained from the PSP measurements. The finalized concept of adiabatic film cooling effectiveness can therefore be written as a function of \(PR\) and \(MW\) (molecular weight ratio between the foreign gas and air = \(\frac{MW_{CO_2}}{MW_{air}}\)) as shown in Equation 4.8 [4, 25].

\[
\eta = 1 - \left[1 + MW \left(\frac{PR_{air}}{PR_{mix}} \bigg|_w - 1\right)\right]^{-1}
\]  
(4.8)

4.5 PSP Processing

For each experiment, a total of 20 background images, reference images, and test images are taken and saved as .tiff files. These images are then imported into Matlab where a code is made to post-process the images to obtain effectiveness values. In Matlab:

- load, crop, and average images
- create \(x/D\) and \(y/D\) vectors
- calculate \(IR\) from Equation 4.5
• calculate $PR$ from Equation 4.7 using the given coefficients from the calibration curve
• calculate local $\eta$ from Equation 4.8
• calculate lateral average effectiveness

4.6 Uncertainty Analysis

The experimental uncertainty was calculated by following the methodology from Figliola and Beasly [12] as well as Natsui et al. [27], whose work goes in detail about PSP uncertainty. The propagation of error of effectiveness values take into account random errors associated with intensity values, pressure, temperature, and the polynomial curve fit from the calibration curve (refer to Figure 4.3).

Uncertainty maps from $\eta$ provide a localized error variation showing lower measurement errors in the region closer to injection and higher values further downstream. These uncertainties vary from 3% up to a local maximum of 10% obtained with a 95% confidence level. Sources of errors from the blowing ratio $M$ calculations include the coolant and freestream temperatures and pressures, with the volumetric flow rate measurements providing the highest contribution (refer to Figure 4.4). A maximum measurement error of up to 5% was obtained for $M$. The uncertainty analysis presented in this section was performed for the lowest blowing ratio of $M = 0.03$ it provides the highest uncertainty.

4.7 Numerical Setup and Boundary Conditions

A reduced model was chosen to represent the entire domain on order to reduce computational time. The computational domain consists of a small section of the test article containing 3 sets of inner and outer hole rows. This section is identically reproduced through the entire domain as the location of the inner lattice with respect to inner and outer hole location repeats in this configuration (refer to Figure 4.6).
The lattice is entirely modeled to avoid losing the flow physics from the structure itself. A multi-component gas model coupled with the SST k-Omega turbulence model is applied using STAR CCM+. The k-Omega model is chosen as it is known to be suitable for resolving the flow features in the viscous sublayer. Flow separation is expected for the current case due to the jets interacting with the cross-flow. In addition, regions of recirculating flow are expected after stagnation occurs at the leading edge.

A periodic boundary condition is applied to both $xz$-planes, while symmetry is claimed at the center of the domain in the $xy$-direction. Air and $CO_2$ are used to model non-reacting species to obtain the desired density variation of $\sim 1.5$; therefore, secondary and cross-flow mass flow boundary conditions are imposed for both inlets from the coolant and mainstream side, respectively. Similarly, an atmospheric pressure outlet boundary condition is implemented to the outlet.
Figure 4.4: Blowing ratio uncertainty tree

Figure 4.5: Repeatability of effectiveness at various blowing ratios: $M = 0.03$, $M = 0.12$, and $M = 0.28$
Figure 4.6: Computation domain and boundary conditions

To follow the experiment as close as possible both temperatures of the coolant and freestream are kept the same in order for the mass transfer analogy to be valid. The computational domain is discretized using an unstructured polyhedral mesh. A mesh sensitivity analysis is provided in Table 4.1 for four different mesh sizes. Using the surface average adiabatic film cooling effectiveness $\bar{\eta}$ as the convergence parameter, a mesh size of 13.17 million cells was selected. The wall $y+$ criteria was kept $< 1.5$ for the leading edge and downstream wall and $< 5$ for the rest of the domain.

4.8 Validation Case

Lateral average effectiveness values for both CFD and experimental results are compared against correlations found in open literature applicable for a blowing ratio of 0.03 (see Figure 4.7).
Table 4.1: Mesh sensitivity analysis

<table>
<thead>
<tr>
<th>Mesh count (10^6)</th>
<th>$\bar{\eta}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>9.4</td>
<td>0.340</td>
</tr>
<tr>
<td>11.0</td>
<td>0.348</td>
</tr>
<tr>
<td>13.17</td>
<td>0.352</td>
</tr>
<tr>
<td>17.06</td>
<td>0.355</td>
</tr>
</tbody>
</table>

Transpiration cooling over a flat plate is experimentally investigated by Goldstein et al [14], while slot-film cooling over a flat plate is also experimentally investigated by Scesa et al. [30]. The streamwise direction is normalized by $M_s$, which is defined as the blowing ratio multiplied by the width of the porous slot (arc length of the transpiration-cooled segment $s = 0.0067m$). Both CFD and experimental values result in very good agreement. However, they slightly under-predict both correlations, specially near injection. This can be partly due to the differences obtained from flow over a leading edge as compared to flat surfaces. Therefore, it is expected to have lower effectiveness results from the current case in comparison to the correlations presented due to the presence of leading edge effects. After stagnation occurs, separation and zones of circulating air lowers the effectiveness of the film. In addition, the test article contains high levels of roughness responsible of lower effectiveness when compared to smooth surfaces [3] (this observation can be valid assuming that the geometries used from the correlations are smoother than for the current case). The correlation from Scesa et al. [30] provides a closer effectiveness trend to the results obtained from both CFD and experiment. Better agreement between the results is obtained as $x/M_s$ moves away from the exit of the porous region. Comparing results from a leading edge to that of a flat plate can cause discrepancy between the correlations and the current study. In addition to that, having different geometries, roughness, porosity, temperature differences, build quality and other testing conditions can also play a role on effectiveness values.
Figure 4.7: Comparison of adiabatic film-cooling effectiveness with correlations found in literature for $M = 0.03$
CHAPTER 5: RESULTS

Adiabatic film cooling effectiveness measurements are obtained using PSP at various blowing ratios ranging between \( M = 0.03 \) and \( M = 0.28 \). For the sake of comparison, two different blowing ratio definitions are used to quantify the mass flux from Equation 1.8. From the transpiration-cooling definition \( (M) \), the area \( A \) is defined as the surface area from the transpiration strip segment \( (A_{\text{Transpiration}}) \), while the film-cooling blowing ratio \( (M_{FC}) \) is defined using the total combined area of the discharge holes \( (A_{\text{Film-Cooling}}) \). Having these concepts will make it easier to draw conclusions by comparing current results to other transpiration-cooling and film-cooling studies due to the particular design of the current test article. The blowing ratio results using the two definitions are found in Table 5.1.

To know the consistency and uniformity of the porous segments and to disregard any asymmetry that can occur from the boundary layer, two different sets of experimental data are taken in plane A and plane B (see Figure 5.1). Plane A is taken towards the center of the test article \((-0.25 < y/D < 0.25)\), while plane B is taken towards the bottom \((-0.14 < y/D < -0.64)\) (Figure 5.1). Local and lateral average effectiveness results are obtained from both planes A and B, respectively, for blowing ratios ranging between \( 0.03 < M < 0.12 \) (refer to Figure 5.2). Agreement between both sets of results is obtained indicating film uniformity from the engineered-porous through the span of the leading edge part. This information is also a good indication of having no edging effects or boundary layer asymmetry affecting the behavior of the flow.

<table>
<thead>
<tr>
<th>Blowing Ratio Definition</th>
<th>(a)</th>
<th>(b)</th>
<th>(c)</th>
<th>(d)</th>
</tr>
</thead>
<tbody>
<tr>
<td>( M = f(A_{\text{Transpiration}}) )</td>
<td>0.03</td>
<td>0.06</td>
<td>0.12</td>
<td>0.28</td>
</tr>
<tr>
<td>( M_{FC} = f(A_{\text{Film-Cooling}}) )</td>
<td>0.21</td>
<td>0.51</td>
<td>1.11</td>
<td>2.39</td>
</tr>
</tbody>
</table>
5.1 Local Adiabatic Film Cooling Effectiveness Comparison of Results

Local effectiveness contours from the experimental measurements are taken for plane B (Figure 5.3) for a relative location of $-0.25 < y'/D < 0.25$ (where $y' = 0$ is at the centerline of each PSP plane). Local effectiveness increases with blowing ratio and decreases along $x/D$. Low effectiveness values and poor coverage is seen for the lowest blowing ratio case $M = 0.03$, where coolant drastically deteriorates before reaching the end of the leading edge. A delay in jet mixing is observed when increasing the blowing ratio, therefore allowing for higher coolant coverage and increased effectiveness values for all cases. Having a delay in mixing is desired, since early flow mixing can be detrimental towards effectiveness, as with improper coolant renewal, the film temperature can soon reach that of the hot stream [9]. In addition, high film uniformity is seen for blowing ratios $M < 0.12$, a beneficial trade expected from transpiration-cooling. On the other hand, spots of discrete cooling is observed for $M = 0.28$ as an indication of jet-lift off.
It is important to point out that the transpiration-cooled design presented in this study is not "purely" transpiration since the engineered-porous region is constrained by sets of inner and outer holes. The reason for having such configuration is to reduce the impact of the design in the structural strength as compared to having an entirety porous structure through. Therefore, it is expected for flow to eventually separate due to the increased momentum experienced by the coolant when passing through the discharging jets. However, the flow momentum is predicted to be much lower than for traditional film cooling due to the pressure loss encountered by the coolant flow due to the lattice.

Local effectiveness contour maps from the numerical simulations are also investigated (see Figure 5.4). At the lowest blowing ratio of $M = 0.03$, CFD effectiveness values slightly over-predict those from the experiment, still showing a rapid decay about half-way of the leading edge. Similar to the experimental results, effectiveness increases with blowing ratio and decreases with $x/D$. Jet lift-off is suspected for $M = 0.12$ and $M = 0.28$, where areas of poor coverage can easily
be spotted. At $M = 0.12$, the point of film re-attachment occurs midway through the curvature of the part while this one is delayed for $M = 0.28$ due to stronger jet separation. It is evident that for the lower blowing ratios of $M = 0.03$ and $M = 0.06$ local effectiveness results are very uniform among the three set of holes, while for the two highest blowing ratios of $M = 0.12$ and $M = 0.28$ there are regions of higher performance than others.

Local effectiveness contour maps from the numerical simulations are examined at $y'/D = 0$ for all blowing ratio cases (see Figure 5.5). At $M = 0.03$ signs of ingestion from the cross-flow and secondary flow interaction is observed attributed to the low coolant flow rates allowing the high momentum free-stream cross-flow penetrate into the engineered-porous cavities. Higher ingestion is obtained from the discharge holes that are closer to the stagnation region. It is known that in GT’s, ingestion from the hot gases is highly detrimental towards its components. As previously observed, very small effectiveness values can be seen at this blowing ratio where coolant propagation is minimal. At $M = 0.06$ there are no signs of freestream ingestion, therefore it is expected not to have any for the rest of the cases as the freestream velocity is kept fixed while the coolant flow rate is increased. For $M = 0.12$, higher effectiveness values hence higher coverage result from the increased coolant flow rate. At this planar location it is difficult to notice the jet separation, however at the highest blowing ratio of $M = 0.28$, the ”jetting” effect is clearly observed due to the higher coolant momentum.

To understand the performance of the discharge holes closely, local numerical effectiveness results are examined at three different planar locations; at $y'/D = 0.04$ some of discharge holes are partially blocked by the lattice, whereas at the planar location of $y'/D = 0$ there is almost a free path between the fluid and the discharge holes and at $y'/D = -0.04$ there is no free path nor blockage from the lattice. Contour maps for all blowing ratio cases are investigated and can be found in Appendix C (refer to Figures C.1, C.2, C.3 and C.4). Very consistent effectiveness values and coolant flow propagation are observed through each plane for $M < 0.12$. 

43
Figure 5.3: Experimental effectiveness distribution at various blowing ratios: (a) $M = 0.03$, (b) $M = 0.06$, (c) $M = 0.12$, (d) $M = 0.28$

For $M = 0.28$ the jet lift-off is clearly seen for all planes; rapid diffusion of the coolant is observed for $y'/D = 0.04$ and $y'/D = 0$ after injection whereas for $y'/D = -0.04$ the jets seem to leave the surface with lower momentum yielding to a stronger re-attachment zone. The lower performing holes from planes $y'/D = 0.04$ and $y'/D = 0$ are due to the velocities experienced by the blocked holes and direct fluid path, respectively as opposed to having neither. However, the given information through each of these planar locations does not take into account its neighboring row-to-row interaction that change the resultant behavior; therefore, it is important to keep this in mind for future analyses.
5.2 Lateral Averaged Effectiveness Comparison of Results

Experimental lateral average effectiveness results are compared against numerical values at different blowing ratios. Similar trends are observed between the numerical and experimental measurements. Higher effectiveness results increase with increasing blowing ratio and monotonically decrease along the stream-wise direction \( x/D \). Early jet lift-off is seen for \( M = 0.12 \) from the numerical case as opposed to the experimental measurements which only show lift-off at the highest blowing ratio. This can be attributed to the CFD under-estimating the viscous losses produced in high velocity areas due to the roughness not accounted for in the numerical analyses. However, better agreement between CFD and the experiment is shown for \( M = 0.28 \), which according to literature, the k-Omega turbulence model has the tendency of over-predicting non-separated flows.
Figure 5.5: Numerical effectiveness distribution at $y'/D = 0$ for various blowing ratios: (a) $M = 0.03$, (b) $M = 0.06$, (c) $M = 0.12$ and (d) $M = 0.28$

and under-predict separated flows.

The point of re-attachment at this blowing ratio is at $\sim x/D = 0.2$ from the experimental case. Numerical results show a delayed flow re-attachment close towards the end of the leading edge
$\sim x/D = 0.3$. Early flow re-attachment from the experiment can be attributed to over-predicting jet velocities from a smooth ideal geometry. It is therefore expected to have lower effectiveness values for regions with higher surface roughness as it is the case for the experimental case.

Surface average film cooling effectiveness is calculated for the area of $-0.25 < y'/D < 0.25$ and $0 < x/D < 5$ (refer to Figure 5.7). An increase in surface average effectiveness is obtained as blowing ratio is increased for both CFD and numerical results. This indicates that although jet separation occurs at $M = 0.28$ for the experimental case, and at $M \geq 0.12$ for the numerical results, the film is still providing higher effectiveness values as compared with the lower blowing ratios.

Figure 5.6: Lateral average effectiveness comparison between experiment and CFD for various blowing ratios
5.3 Discharge Coefficient

Another parameter of interest is the discharge coefficient $C_d$, which can provide quantitative measurements of the quality of the transpiration-cooling geometry. This definition, found in Equation 5.2, is a ratio of the actual mass flow rate divided by the ideal. $C_d$ is calculated assuming equal mass flow rate going through every hole. The region utilized for the $C_d$ calculations is the entire porous section, which includes the inlet and outlet holes as well as the lattice. To obtain the ideal mass flow rate ($\dot{m}_{\text{actual}}$) from the experiment, the total combined designed area of the discharge holes is used instead of the actual total combined area. The reason for this is due to the lack of information and ability to measure the actual area. Other quantities used are the stagnation pressures from the upstream ($P_c$) and downstream ($P_\infty$) regions of the porous segment; $P_c$ is directly measured by a static pressure tap placed upstream the porous region, while $P_\infty$ is taken
from the side wall of the discharge plenum assuming this will provide a good representation of the discharge pressure.

Experimental discharge coefficient results are compared against numerical values (refer to Figure 5.8). Experimental $C_d$ results show a decreasing trend as the jet Reynolds number $Re_d$ is increased. Replicating the same pressure measurements in CFD leads to similar trends while over-predicting experimental results by about 25% (refer to the CFD case in Figure 5.8). If the actual area for the experimental calculations is used, the $C_d$ values are expected to increase because the actual area is predicted to be smaller than the ideal area due to the imperfections associated with the manufacturing technique as previously mentioned in the NDE section. A second set of numerical $C_d$ results is also plotted along the other cases (refer to the CFD* case in Figure 5.8). In this case the true stagnation pressures upstream and downstream of the porous cavities is utilized therefore providing an increasing trend as $Re_d$ increases. The decreasing behavior from the experimental results is suspected to be attributed to not properly capturing the true stagnation pressures specially at the leading edge. This is most likely related to the differences in pressure coefficients experienced by flow over a cylindrical geometry. What this signifies is that the current experimental setup does not allow for the correct measurements of the stagnation properties representative of the coefficient of discharge equation. For better characterization of the discharge behavior for this particular geometry, another setup using the same porous structures must be utilized where upstream and downstream true stagnation pressures can be measured.

A better metric to account for losses to describe the quality of the part is by calculating the pressure drop across the porous region. These results are plotted at various Reynolds numbers (see Figure 5.9). Results show that CFD under-predicts experimental values of up to 50% observed at the highest $Re_d$. This is expected due the high effective roughness encountered by the actual geometry that is not accounted for in the CFD simulations. This difference increases with Reynolds number due to the viscous losses that are more dominant in regions of higher velocities. These results are significant as this tells us that adjustments in the CFD models need to be used in order
to better predict the behavior of SLM parts as this is unable to characterize the true geometrical shape which provides drastic changes in the flow parameters.

\[
Re_d = \frac{\dot{m}_c d}{\mu A_{Film-Cooling}} \quad (5.1)
\]

\[
Cd = \frac{\dot{m}_{actual}}{A_t P_c (\frac{P_\infty}{P_c})^{\frac{\gamma+1}{2\gamma}} \sqrt{\frac{2\gamma}{\gamma-1}} \frac{1}{RT_c} \left[(\frac{P_c}{P_\infty})^{\frac{\gamma-1}{\gamma}} - 1\right]} \quad (5.2)
\]

Figure 5.8: Discharge coefficient comparison
Figure 5.9: Pressure drop comparison
CHAPTER 6: FUTURE WORK

This section emphasizes one of the many possible routes that can be taken to extend this work using transpiration cooling. As an assessment to take into account the integrity of the transpiration-cooled geometries, a new geometry has been designed incorporating the lattice cooling geometry as well as other aspects when it comes to manufacturing. This section is dedicated to a new designed developed at DLR (German Aerospace Center) in conjunction with UCF. The same transpiration-cooling idea mentioned in this study was taken and implemented into a “dog-bone” tensile specimen in order to perform structural testing. This would allow us to understand how feasible this part is due to the unconventional cooling geometry and material itself under real-engine conditions. This part is to be manufactured at DLR (German Aerospace Center) using an SLM machine from Inconel 718 powders. The tensile specimen was designed not only to meet cooling-design specifications but also manufacturing considerations from the LAM/SLM technique at DLR (refer to Figure 6.1).

6.1 Manufacturing Considerations

There is huge compromise between design limitations, part quality and surface roughness. Therefore, it is common for manufacturing engineers to look for the most optimal way of fabrication if the design allows it. The build direction of this part is in the upward direction. This direction is optimal for this particular design, and because of this, the orientation of the holes and lattice were slightly modified in comparison to the original design. The lattice was designed in a slanted-manner with respect to the build direction in order to avoid any unnecessary supporting frames (see Figure 6.2). An option to changing the lattice design and avoid supporting frames would have been to build the part at an angle instead of an upright direction. However, it has been found that building the part at an angle can affect both the part’s quality and surface roughness.
Figure 6.1: LAM tensile specimen

Figure 6.2: Tensile specimen inner lattice design
The surface roughness introduced from additively manufacturing parts is inevitable, and vast amount of research has been done to control it and quantify it. This roughness is random and inability to control it can lead to many issues and compromise the material’s integrity. Roughness can appear in the form of surface roughness, material residues from the supporting frames, and other problems encountered from the manufacturing technique. More information on build-quality of SLM/LAM parts is investigated by Ealy et al. [6] and Snyder et al. [31]. Other SLM design considerations taken for this particular designed are also described (see Figure 6.3).

Figure 6.3: Tensile specimen structure
6.2 DLR Testing Goals

The number of holes and patterned-lattice sections can be adjusted in order to meet certain flow requirements for experimental purposes. The test specimen will be additively manufactured at the DLR facilities and will be used to perform stress-strain measurements using X-Ray Diffraction (XRD) and Digital Image Correlation (DIC). In the experimental setup, thermal and mechanical loading will be applied to the specimen in order to achieve real engine conditions. The thermal loading will be provided in the form of radiation, and cooling will be induced inside of the part causing thermal gradients responsible for high thermal stresses. The temperature field will be captured by an infrared (IR) camera to obtain local surface temperature that can be mapped along the measured stress/strain values (refer to Figure 6.5).
Figure 6.5: Schematic of the experimental setup
CHAPTER 7: CONCLUSION

Transpiration cooling is a promising technology capable of reducing the heat transfer from the hot freestream gases to the surfaces of blades or vanes in gas turbine applications. It has been found that film cooling uniformity as well as low coolant flow-rate consumption are some of the advantages of such cooling technique. The fabrication of transpiration-cooled components for IGT’s is not possible through conventional casting but through the use of LAM techniques. In the present study, an engineered-porous mock leading edge segment of a turbine blade fabricated from LAM is investigated. This component, manufactured from Inconel 718 powders, contains an engineered-lattice structure used to simulate the transpiration-cooling effect. Adiabatic film cooling effectiveness is experimentally and numerically investigated for blowing ratios ranging between $M = 0.03$ and $M = 0.28$. Effectiveness values are obtained from the mass and heat transfer analogy, where a foreign fluid is used as the secondary flow. To achieve a nominal density ratio of 1.5, $CO_2$ is utilized as the coolant flow, while air is used as the freestream cross-flow.

Effectiveness distributions are compared for both CFD and experimental measurements. Experimental results display high film uniformity up to the blowing ratio of $M = 0.12$, while areas of discrete cooling is spotted for $M = 28$ as an indication of jet lift-off. On the other hand, numerical results indicate to have film uniformity for $M = 0.03$ and $M = 0.06$, while flow seems to lift-off for $M = 0.12$ and $M = 0.28$. Regions of high lift-off is evident at the highest blowing ratio where areas of poor effectiveness are noticed downstream injection. While jet separation can be detrimental, surface average results are the highest at the higher blowing ratios.

Lateral average effectiveness results are also investigated for both CFD and experiment. A slight over-prediction in effectiveness is observed from CFD at the curvature region for $M \leq 0.12$. Better agreement is seen for $M = 0.28$ and as $x/D$ increases for all the cases. Jet lift-off is shown for $M = 0.12$ and $M = 0.28$ for the numerical case while it is only seen at the highest blowing ratio of $M = 0.28$ from the experimental measurements. Upon further examination of
the numerical results, ingestion from the freestream is only seen for the lowest blowing ratio of \( M = 0.03 \) due to the low momentum coolant flow rate. However, no signs of ingestion is obtained as the blowing ration increases since the freetream velocity is kept constant.

Through a NDE assessment, it is clear how much the actual part deviates from the ideal geometry due to the manufacturing tolerances offered from the manufacturing technique. Build quality and characteristics such as imperfections and material residue are some of the many important parameters to consider that affect the flow behavior and ultimately the heat transfer. CFD can be used as a tool to estimate the performance of LAM/SLM parts, however, inability to account for these imperfections or what is refereed earlier to as ”effective roughness” can bring inaccuracies into the numerical models. This example can be seen from the pressure drop results obtained from CFD, which resulted in an under-prediction from the experimental values of up to 50% at the highest Reynolds number. This is because a perfectly smooth and ideal geometry is what is modeled in CFD instead of the actual part. Therefore, we need to find ways to better predict the behavior of SLM parts in numerical simulations by not just including the roughness but also being able to quantify and model the actual part as close as possible.
Figure A.1: $CO_2$ Setup

Figure A.2: Experimental Setup
APPENDIX B: MESH SCENES
Figure B.1: Mesh cross-section scene

Figure B.2: Mesh scene for different cross-sections
Figure B.3: Cell surface cross-section scene
APPENDIX C: NUMERICAL RESULTS AT DIFFERENT PLANAR LOCATIONS
Figure C.1: Numerical effectiveness distribution at $M = 0.03$ for various planar locations: (a) $y'/D = 0.04$ (b) $y'/D = 0$, (c) $y'/D = -0.04$

Figure C.2: Numerical effectiveness distribution at $M = 0.06$ for various planar locations: (a) $y'/D = 0.04$ (b) $y'/D = 0$, (c) $y'/D = -0.04$
Figure C.3: Numerical effectiveness distribution at $M = 0.12$ for various planar locations: (a) $y'/D = 0.04$ (b) $y'/D = 0$, (c) $y'/D = -0.04$

Figure C.4: Numerical effectiveness distribution at $M = 0.28$ for various planar locations: (a) $y'/D = 0.04$ (b) $y'/D = 0$, (c) $y'/D = -0.04$
LIST OF REFERENCES


