5-2016

Analysis of a Coupled Micro- and Triple-Impingement Cooling Configuration in the C3X Vane

Chase D. Rossman

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

Part of the [Aerospace Engineering Commons](#)

Scholarly Commons Citation

Rossman, Chase D., "Analysis of a Coupled Micro- and Triple-Impingement Cooling Configuration in the C3X Vane" (2016). *Dissertations and Theses*. 238.
<https://commons.erau.edu/edt/238>

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.

ANALYSIS OF A COUPLED MICRO- AND TRIPLE- IMPINGEMENT COOLING
CONFIGURATION IN THE C3X VANE

A Thesis
Submitted to the Faculty of
Embry-Riddle Aeronautical University

by

Chase D. Rossman

In Partial Fulfillment of the
Requirements for the Degree of
Master of Science in Aerospace Engineering

May 2016
Embry-Riddle Aeronautical University
Daytona Beach, Florida

ANALYSIS OF A COUPLED MICRO- AND TRIPLE- IMPINGEMENT COOLING
CONFIGURATION IN THE C3X VANE

by

Chase D. Rossman

A Thesis prepared under the direction of the candidate's committee chairman, Dr. Mark Ricklick, Assistant Professor, Department of Aerospace Engineering, and thesis committee members Dr. Sandra Boetcher, Assistant Professor, and Dr. Magdy Attia, Professor, 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

Chairman, Dr. Mark Ricklick

Member, Dr. Sandra Boetcher

Member, Dr. Magdy Attia

Department Chair, Dr. Anastasios Lyrantzis
or Graduate Program Coordinator, Dr. Magdy Attia

Date

Dean of College of Engineering, Dr. Maj Mirmirani

Date

Associate VP for Academics, Dr. Chris Grant

Date

ACKNOWLEDGMENTS

First and foremost I would like to acknowledge all the information, guidance, and support of my thesis chairman Dr. Mark Ricklick who has been instrumental in the overall development and molding of my way of thinking for the industry ahead. I cannot express how grateful I am to have had an advisor who values his role as an educator as highly as Dr. Ricklick does. Without his willingness to oversee and provide project recommendations throughout the projects research, this projects completion wouldn't have succeeded.

Next I would like to extend a thank you to the thesis committee members Dr. Attia and Dr. Sandra Boetcher. Both instructors helped highlight two areas of the aerospace industry that I wanted to explore more about, making the classes taught by these instructors the ultimate reason to pursue a Master's degree at Embry-Riddle Aeronautical University pertaining to heat transfer in gas turbine applications. I also would like to extend a thank you for the life lessons given in class, making topics fun to learn, and applying them within the pursuit of a master's degree. I would also like to recognize the committee for their comments regarding this thesis and am grateful for the effort you put forth as members of my committee.

Finally I would like to dedicate this work to my academic peers, personal friends, and most importantly my family members that have helped push me in the pursuit for my master's degree. Whether it is from late nights to just simple motivation, I know that success at this time in my life wouldn't have happened without the continued support.

TABLE OF CONTENTS

LIST OF TABLES	vi
LIST OF FIGURES	vii
SYMBOLS.....	x
ABBREVIATIONS	xi
NOMENCLATURE	xii
ABSTRACT.....	xiii
1. Introduction.....	1
1.1. Brayton Cycle	2
2. Literature Review	7
2.1 Turbine Blade Cooling	7
2.1.1. Trailing Edge Cooling Concepts	11
2.1.2. Jet Impingement Cooling.....	17
2.1.3. Micro-Cooling Channels.....	21
2.2. Conjugate Heat Transfer	27
2.3. Objective Statement.....	34
3. Methodology.....	35
3.1. Introduction	35
3.2. Baseline Domain Setup	35
3.2.1. Continua Setup	41
3.2.2. Turbulence Models.....	47
3.2.3. Conjugate Heat Transfer	51
3.3. Micro-Cooling Case Specifics	53
3.4. Triple Jet Impingement Combination Specifics	58
4. Data Reduction	62
4.1. Introduction	62
4.2. Model Variations and Monitoring	62
4.3. Heat Transfer Reduction	67
4.4. Domain Flow Validation.....	70
5. Results.....	74
5.1. Effects of Cooling.....	74
5.1.1. Baseline Configuration Calibration.....	74
5.1.2. Micro-Cooling Configuration Calibration	77
5.1.3. Jet Impingement Configuration Comparison	80
5.1.4. Trailing Edge Focus	83
5.2. Heat Transfer of Cooling Channels	85
5.2.1. Triple Impingement Performance Investigation.....	91
5.3. Cooling Performance.....	97

5.3.1.	Cooling Effectiveness	101
6.	Conclusion	103
7.	Recommendations.....	105
REFERENCES		106
APPENDICES		115
A.	C3X Vane Coordinate.....	115
B.	Flow Field Contours	116
1.	Current Study Baseline Configuration Flow Contours:	116
1.1.1.	Current Study Baseline Configuration Cascade Flow Contours:.....	118
1.2.	Current Study Micro-Cooling Configuration Flow Contour	120
1.2.1.	Current Study Micro-Cooling Configuration Cascade Flow Contours:	122
1.3.	Current Study Triple-Impingement Configuration Flow Contours:	124
1.3.1.	Current Study Triple-Impingement Configuration Cascade Flow Contours:	126

LIST OF TABLES

Table 1: Turbulence Models Utilized in Wang's Experiment with Characteristics	23
Table 2: Initial Cooling Channel Input Dimensions and Initial Cut Domain Fluid Flow Conditions	39
Table 3: Final Mesh Specifications.....	41
Table 4: Micro-Cooling Channel Inferred and Calculated Input Dimensions.....	55
Table 5: Calculated Cooling Performance Parameters	97
Table 6: Calculated Cooling Effectiveness	101

LIST OF FIGURES

Figure 1: Real World Example of Brayton Cycle and Ideal Isentropic Thermodynamic diagrams. (Brayton Cycle)	2
Figure 2: 40 years of Jet engine Designs compared to Overall Pressure ratio of the engine (Janes Aero-engines 1998 ref. (Gunston, 2001))	4
Figure 3: Graphical representations of the TIT and its relation to engine performance per cooling (a.) and then core power (b.) (Boyce, 1982)	7
Figure 4: Various Suggested cooling schemes being researched or implemented. (Boyce, 1982)	9
Figure 5: (Halila, 1982) experiment on Energy Efficient Engine (E^3) showing various cooling concepts for the 1 st Stage High pressure turbine vane	11
Figure 6: (Halila, 1982) experiment on Energy Efficient Engine (E^3) showing the varying temperature for the 1 st Stage High pressure turbine vane	12
Figure 7: Comparative Image of two different trailing edge cooling concepts a.) weave b.) triple impingement (Weaver, 2015)	15
Figure 8: Side view of a three cavity structure in a Turbine trailing edge (Chyu, 2010)	15
Figure 9: Flow regions of a single impinging jet a) complete jet structure and typical region development b) is the core development of the jet. (Han, 2013).....	17
Figure 10: Square Jet Array utilized by (Yamane, 2012) to clarify Jet Array impingement heat transfer.....	19
Figure 11: Industry Standard view of turbine blade/vane Inside Serpentine Shape Cooling Configuration. (Chyu, 2010).....	21
Figure 12: One of (Dennis, 2003)'s optimization cases using PGA optimizer a.) Baseline case b.) PGA optimizer configuration	25
Figure 13: Rig Layout used by (Coletti, 2012) for rib CHT analysis a.) Full rig with highlighted sections b.) Rib setup	30
Figure 14: Schematic of Airfoil used by (Dees, 2010) and a sample of model results a.) Airfoil b.) Film cooling for conductive airfoil c.) Adiabatic airfoil effectiveness	31
Figure 15: C3X finite element grid structure with cooling hole locations shown. (Hylton, 1983)	35
Figure 16: 2.52 cm baseline C3X turbine guide vane replication.	36
Figure 17: Model Setup (Subdomains: hot flow, vane, coolant)	37
Figure 18: Reynolds Number comparison of Baseline cases for different studies	40
Figure 19: Basic visual Mesh Specifications: a) surface curvature, b) Number of prism layers, c) skewness angle, d) decreasing base size (CD-adapco, 2014)	42
Figure 20: Graph Representation of Mesh Independence Study with Percent differences	43

Figure 21: Transparent View of Meshed C3X Baseline Channel Model with Mesh Refinements	44
Figure 22: Histogram displaying y^+ wall treatment per cooling channel.....	45
Figure 23: Display showing the extent of three methodologies used to solve turbulent flows.....	49
Figure 24: a) Baseline Vane configuration b) Micro-cooling configuration (Highlighted Super ellipse dimensions)	53
Figure 25: Baseline Case (a) and micro-cooling case (b) showing regions used to interpret Wang's Experiment temperature and coolant distribution for micro-cooling channels.....	56
Figure 26: Inlet Reynolds Number comparison of C.S. Micro-cooling and Triple Impinging Configuration.....	57
Figure 27: Isometric Cut View of Dual cooling configuration (Highlighted Trailing Edge Triple Impingement Jet Locations).....	58
Figure 28: Exit Mach number comparison of C.S. Micro-cooling and Triple Impinging Configuration	60
Figure 29: Graph displaying Monitor plot of activated fully developed interface	63
Figure 30: Diverged Mole results for C.S. baseline case highlighting a) Various Temperature monitors and b) high pockets of mass flux.....	65
Figure 31: Top Views of Different Flow Conditions of Baseline Case produced by Current model	70
Figure 32: C3X vane Normalized Span Coordinate Orientations used for data reduction (a) Whole Vane (b) Trailing Edge	73
Figure 33: Current Study Mid-span data compared to C3X vane data for different flow characteristics.....	75
Figure 34: Top View Comparison of the Baseline Configuration.....	76
Figure 35: Micro-Cooling CHT measured values	77
Figure 36: Top and Side View of the Micro- Cooling Configuration	78
Figure 37: Current study CHT measured values for various vane configurations.....	80
Figure 38: Top Views Comparing the Temperature distribution of the three configurations produced in current study	81
Figure 39: Highlighted Side Views of the back 1/3 rd Temperature contours of the three configurations produced in current study	83
Figure 40: Current study Trailing Edge Temperature Data for various vane configurations	84
Figure 41: Comparison of Heat Transfer Coefficient of Cooling Channels for Baseline data.....	85

Figure 42: Comparison of the average HTC Cooling Channels for C.S. Micro-Cooling and Triple Impingement Configurations.....	86
Figure 43: Triple Impingement Channel HTC Contour Plots a) full vane b) highlighted impingement channels with different contour scale	88
Figure 44: Triple Impingement Channel Temperature Contour Plot	89
Figure 45: Side View of Vane Showing Cooling Channel Velocity Vectors.....	91
Figure 46: Temperature Vectors of Triple Impingement a) full configuration b) highlighted vectors between 1 st and 2 nd triple impingement channel	92
Figure 47: Section view of Triple Impingement Jet velocity Magnitude (Highlighted Trailing Edge Ejection).....	93
Figure 48: Triple Impingement Ejection Jet with recirculation zone	94
Figure 49: Section view of Triple Impingement Jet velocity Magnitude with larger pressure difference (Highlighted Trailing Edge Ejection).....	95
Figure 50: Temperature Vectors of Triple Impingement full configuration with larger pressure difference	96
Figure 51: Trailing Edge Vane channel location view a) baseline b) Micro-cooling configuration and c) Triple- impingement configuration	102

SYMBOLS

Greek Symbols

ρ_{ss}	Density of Stainless Steel (kg/m ³)
ρ_{sf}	Density of Fluid (kg/m ³)
λ	Thermal Conductivity (W/(m·K))
μ	Fluid Viscosity (Pa·s)
μ_t / μ	Eddy Turbulent Viscosity
β and β^*	SST k- ω Turbulence Model Coefficient
γ_{eff}	SST k- ω Turbulence Effective Intermittency
μ_t	SST k- ω Relation for Turbulent Viscosity
η_{tt}	Total to total turbine efficiency

Symbols

D_c	Cooling Channel Hydraulic Diameter (mm)
D_J	Jet Hydraulic Diameter (mm)
S_{jj}	Jet-to-jet Spacing (mm)
K_{fg}	Thermal Conductivity of Fluid (W/(m·K))
T_0	Static Temperature (K)
T_i	Total Temperature (K)
P_i	Total Pressure (Pa)
P_0	Static Temperature (Pa)
C_{sf}	Specific heat capacity of Fluid (J·K/kg)
C_{ss}	Specific heat capacity of Stainless Steel (J·K/kg)
K_{ss}	Thermal Conductivity of Stainless Steel (W/(m·K))
Re	Reynolds Number
Nu	Nusselt Number
H	Length (mm)
D_ω	SST k- ω Turbulence Cross-Derivative
G_k	SST k- ω Turbulent Production
S_k and S_ω	SST k- ω Turbulence user-specified source terms
k_0 and ω_0	SST k- ω Turbulence ambient values source terms
Re_D	Hydraulic Diameter Reynolds Number
h_0	NASA reference heat transfer coefficient (1135 W/m ² -K)
h	external airfoil heat transfer coefficient ($q_s'' / (T_s - T)$) (W/m ² -K)
S	Sutherlands Constant
$\dot{m}$	Mass flow rate (kg/s)

ABBREVIATIONS

<i>SST</i>	Shear Stress Transport
<i>CHT</i>	Conjugate Heat Transfer
<i>CFD</i>	Computational Fluid Dynamics
<i>RANS</i>	Reynolds Average Navier Stokes
<i>LES</i>	Large Eddy Simulation
<i>DES</i>	Direct Numerical Simulation
<i>CFL</i>	Courant-Friedrichs-Levy Number
<i>BC</i>	Boundary Condition
<i>CIES</i>	Coupled Implicit Energy Solver
<i>E³</i>	Energy Efficient Engine
<i>HPT</i>	High Pressure Turbine
<i>TIT</i>	Turbine Inlet Temperature
<i>C.S.</i>	Current Study/Studies
<i>HTC</i>	Heat Transfer Coefficient
<i>P.S.</i>	Pressure Side
<i>S.S.</i>	Suction Side

NOMENCLATURE

mm	millimeters
K	Degree Kelvin
Pa	Pascal
$J \cdot K/kg$	Joules Kelvin per Kilogram
$W/m \cdot K$	Watts per Meter Kelvin
$Pa \cdot s$	Pascal per Second
kg/m^3	Kilograms per Meter cubed

ABSTRACT

Researcher: Chase D. Rossman

Title: Analysis of a Coupled Micro- And Triple- Impingement Cooling Configuration in the C3X Vane

Institution: Embry-Riddle Aeronautical University

Degree: Master of Science in Aerospace Engineering

Year: 2016

This study attempts to improve the trailing edge protection of a turbine vane by incorporating two concepts from the literature. These two concepts are optimized near wall cooling passages and a triple-impingement configuration in the vane trailing edge. Finally this study explores a CFD conjugate analysis based on the Shear Stress Transport $k-\omega$ turbulence model. The overall goal is to produce a more effective cooling configuration by studying the effects of the flow and heat transfer of the dual configuration, while keeping the integrity of the turbine vane. Star-CCM+ was utilized to carry out the computational data on a 3-D model of the NASA C3X turbine vane. The analysis compared the temperature and the overall heat transfer coefficient to several data sets from the literature at the vane mid span with a heavy emphasis on the trailing edge portion of turbine vane. Results are generated to show the nature of the flow induced by the coupled system and the thermomechanical performance and the efficiency of the turbine vane. The triple impingement configuration displayed a cooler average temperature distribution of 18.8% than the original NASA C3X vane and a slightly cooler temperature of 2.2% than the original micro-cooling configuration from the literature. Strictly the back 1/3rd of the vane for the triple impingement configuration produced a

2.8%-3.3% cooler average temperature than the original micro-cooling configuration and a 14.7%-18.95% cooler vane than the original NASA C3X vane. The results have the likely hood to produce a tremendous cost savings for an industry utilizing cooling configurations for turbine application.

1. Introduction

In today's gas turbines, the temperatures significantly exceed most materials allowable temperatures of operation. The overall concern is the efficiency of the turbine system as a whole. In order to achieve the necessary temperatures and efficiencies to maintain the integrity, turbine blade cooling is implemented. However with the addition of cooling the difficulties lie in cooling the edges of the turbine blade. The coolant must be effective, satisfying the temperature deemed operable in a turbine, and efficient, utilizing minimal coolant. The edges most often focused on are the leading and the trailing edges of the airfoil, the tip, and the end walls of the turbine. To accomplish the desired increase of temperature in a turbine to produce the most desirable turbine system, the temperature and stress of the blade need to be taken into account simultaneously (Wang, 2015). One way to incorporate these new challenges can be seen by numerous cooling configurations, optimization methods, and extensive conjugate heat transfer (CHT) analyses that have been implemented to test the overall cooling capabilities of heat transfer in turbine blades (Wang, 2015). The overall need of cooling of the turbine region is displayed by the direct forms of cooling techniques utilized in industry such as, impingement cooling, pin-fin cooling, film cooling, and transpiration cooling configurations. These particular cooling configurations have made drastic changes and extreme impacts on the performance capabilities of turbine engines allowing for the turbine inlet temperatures to be pushed higher than material limits.

1.1. Brayton Cycle

The performance capabilities of the turbine engine can be seen in the Brayton Cycle as an ideal isentropic thermodynamic process. Figure 1 is a simple schematic of the Brayton Cycle, the P-V diagram, and the T-S diagram of the isentropic process. The cycle can be seen by the adiabatic process, 1-2, or compression of the air in the system.

Figure 1: Real World Example of Brayton Cycle and Ideal Isentropic Thermodynamic diagrams. (Brayton Cycle)

Followed by the combustion, 2-3, which is heat addition. Then another adiabatic process of expansion, 3-4, through the turbine producing a work output. Finally through isobaric heat extraction, 4, as the exhaust gases exit the cycle (Brayton Cycle).

Even with the Brayton Cycle's beneficial past and future within the gas turbine industry it still has some performance inefficiencies, such as output losses, adverse conditions, parasitic losses, mechanical efficiency, and compressor fouling (Brayton Cycle). An estimated 70 to 80% of all gas turbine performance losses due to fouling have been accumulated during operation (Mehyer-Homji, 2004). Fouling can cause deposits on blades and even move the compressor surge line closer to operating conditions, more on fouling can be found by the author published in 1992. According to the author these particular losses directly affect the airflow, pressure ratio, compressor efficiency, and ultimately affect the power output and thermal efficiency. The net work, $\mathcal{W}_{\text{net}}$, is shown

by subtracting the work from the compressor, $\mathcal{W}_{comp}$, from the work of the turbine, $\mathcal{W}_{turb}$.

The net work produced by the Ideal Brayton Cycle can be seen in Eq. (1):

$$\mathcal{W}_{net} = \mathcal{W}_{turb} - \mathcal{W}_{Comp} \rightarrow C_p(T_3 - T_4) - C_p(T_2 - T_1) \quad (1)$$

Where C_p is the specific heat of the gas and T_x represents the temperatures at the various stations along the Brayton Cycle (Boles). One important parameter that is usually studied throughout experiments is the total pressure drop across a stage. This total pressure drop allows for the stage efficiency to be calculated. With a better stage efficiency across the turbine vane a boost in overall system efficiency can be obtained. The ideal thermodynamic efficiency, η_{th} , is represented in Eq. (2):

$$\eta_{th} = 1 - \frac{1}{r_p^{\frac{\gamma-1}{\gamma}}} \quad (2)$$

Where γ is the specific heat of the gas and r_p is the pressure ratio (p_2/p_1) representing the corresponding stations within the Brayton Cycle (Cohen, 1987). It is evident that Eq. (2) displays that an increase in the compressor pressure ratio increases the thermal efficiency of the cycle. However to see the direct effects of the turbine and compressor the total efficiency of the cycle can be calculated. The real cycle efficiency takes into account the effects of the compress, η_c , the efficiency of the turbine, η_t , the firing temperature, T_f , and ambient temperature of the turbine, T_{amb} and can be seen in Eq. (3) (Boyce,1982):

$$\eta_{real} = \left(\frac{\eta_t T_f - \frac{T_{amb} r_p^{\left(\frac{\gamma-1}{\gamma}\right)}}{\eta_c}}{T_f - T_{amb} - T_{amb} \left(\frac{r_p^{\left(\frac{\gamma-1}{\gamma}\right)} - 1}{\eta_c} \right)} \right) (\eta_{th}) \quad (3)$$

Boosting the overall efficiency of the cycle increases the pressure ratio of a cycle, having a greater impact on the temperature of the core mass flow rate that the

turbine sees exiting the combustion chamber. This can be shown in Fig. 2 below that displays how the overall pressure ratio relates to the engines produced over a span of 40 years.

Figure 2: 40 years of Jet engine Designs compared to Overall Pressure ratio of the engine (Janes Aero-engines 1998 ref. (Gunston, 2001))

The compressor ratio and the ratio of the turbine to compressor inlet temperatures become the utmost important design parameters in gas turbine work (Potter, 1972). The temperature exiting the combustion chamber can't exceed the metallurgical limits of the turbine or catastrophic failure can occur. Multiple avenues have been explored to try and produce a better ideal pressure ratio without exceeding the turbine material temperature limits. Some of these have been intercooling, reheating the flow, the regenerative cycle, and increasing the pressure ratio. The desired effect of utilizing intercooling and reheating is to increase the power of the system as a whole.

Intercooling allows for the compressor to be separated into multiple stages with intercooling acting in between the stages. This introduces two or more adiabatic processes with intercooling and an increase in the next work of the cycle without an increase in the turbine work (Boyce, 1994). The author then introduces reheating the cycle, which results in increasing the turbine work without changing the compressor work or turbine inlet temperature (TIT). This is the idea of splitting the turbine into multi-stages with constant pressure heating before each expansion. The author implies that if both intercooling and reheating is applied to the system the adiabatic thermal efficiency, represented by Eq. (2), is lowered while the work output is increased.

To improve the efficiency of the Brayton cycle both increasing the pressure ratio of the compressor and the regenerative cycle are utilized. Increasing the pressure ratio has some major draw backs such as more leakage flow, an increase in compressor exit temperature, and a drop of compressor efficiency (Boyce, 2000). The author states that increasing the compressor exit temperatures increases the TIT. Thus the implantation of turbine cooling is taken into account to mitigate the TIT while maintaining a high compressor pressure ratio. Finally the prominent method today, is utilizing the regenerative cycle, which helps reduced the amount of fuel needed for the engine to operate (Chodkiewicz, 2000). The author explains that the regenerative cycle uses a regenerator which utilizes the hot turbine exhaust gases to preheat the air between the compressor and the combustion chamber. This increases the air temperature entering the burner and reduces the overall fuel-to-air ratio needed to obtain combustion. The author summarizes that the maximum thermal efficiency for the regenerative cycle occurs at lower pressure ratios than that of the simple cycle.

The total to total turbine efficiency, η_{tt} , allows for the exhaust kinetic energy to be considered in the efficiency calculation since the exhaust gases are intended to emerge at high velocities for turbojets (Hill, 1965). This higher turbine stage efficiency starts to mitigate compressor performance inefficiencies. The Author states these compressor inefficiencies can directly affect kinetic energy in the exhaust of an individual cycle produces while a turbojet is operational. This shaft power, kinetic energy, produced is based off a coupled system with the amount of work a turbine stage can produce over a single stage or a multistage turbine. Outlined by the author the amount of work, w_t , can be calculated from corresponding inlet and outlet conditions and be written utilizing η_{tt} .

In this thesis, a 3-D model of the NASA C3X turbine vane has been developed to implement and establish a combined cooling configuration. This thesis also utilizes a conjugate heat transfer analyses in a turbine vane to increase the efficiency of the turbine stage and the overall efficiency for a turbine engine. This will be accomplished through a comparison of previous experimental data for cooling configurations for turbine blades while exploring CFD CHT analysis. Once the NASA C3X model experimental data (Hylton, 1983) is reasonably matched the micro-cooling channel configuration from Wang's data will be incorporated (Wang, 2015). The final step will be taking the Wang's micro-cooling channel configuration and implementing a triple jet impingement array to produce evidence of enhanced performance characteristics and desirable heat transfer properties (Wang, 2015).

2. Literature Review

This section of the report will walk through the varying parts taken into account while research was undergone, the simulation setup, and fundamental understanding of the varying concepts and experiments to produce the results.

2.1 Turbine Blade Cooling

With the turbine inlet temperatures climbing considerably over the last few

decades it is predicted that the trend will only continue to advance, made possible by advancements in turbine blade cooling technologies (Boyce, 1982). Figure 3 (a) is a graphical representation depicting the advances of turbine cooling over the past 50 years and the increased TIT. This is critical to a turbine based engine because with an increased turbine entrance temperature there is a drastic improvement in the cycle output power, also graphically shown in Fig.3 (b).

These turbine advances have increased the turbines first stage rotor and stator capabilities to withstand the severe temperature, stress, and environment combinations. The increased capabilities allow for an improved fatigue lifetime of the individual component and enhance the output power of the machine as a whole. According to Boyce the level of performance of turbine blades have been brought about by a combination of the following:

- Advances in air-cooling for turbine blades and vanes
- Design Technology for stresses and airflow
- Single Crystal (SC) and directional solidified (DS) casting process improvements
- Development and use of nickel-based super alloys with thermal barrier coatings (Boyce, 1982)

Shown in Fig. 4 are some of the most common cooling schemes being research and implemented in the gas turbine industry. Film cooling allows for the coolant to be discharged through small holes in the blade walls, providing a thin, cool, insulating buffer protecting against the hot gas thermal loads. Convection cooling applies a cooling stream which extracts heat from the blade surface forming a repeating convection current. Full coverage film cooling takes into account varying hole to hole spacing and injection angels of arrays utilized to spread the coolant across the blade surface (Hodges, 2012). Impingement cooling happens within the blades as coolant is forced through the internal

Figure 4: Various Suggested cooling schemes being researched or implemented. (Boyce, 1982)

wall, forcing the flow to impinge on the outside wall of the blade. Finally, transpiration cooling is taking a porous material and passing coolant through the material to extract heat from the intended surface into the cross flow in order to obtain the desired surface temperature (Han, 2013).

Each of these particular cooling schemes has demonstrated enhancement of the efficiency of a turbine blade as a whole or a particular area of the rotor or stator. For example the film cooling scheme has been implemented at the leading edge of most blade configurations to help mitigate the temperature distribution caused by the stagnation region and gas temperature of incoming airflow. This is done by injecting the lower temperature gas into the mainstream gas allowing for the heat transfer into the airfoil to be reduced along with the scheme's other ability is to cover the rest of the rotor or stator with the coolant (Bogard). The fundamental concept of a turbine cooling system is to

maximize the benefits of the inlet gas temperature while minimizing the amount of coolant stolen as the bleed air from the compressor section (Han, 2013). In order to capture the coolants effects on the system various parameters are studied and calculated. The most common of these are the total of the mass flow rates, cooling efficiency, cooling effectiveness , pattern factor, film cooling effectiveness, and heat load parameter just to name a few. A prior experiment conducted by Sivasgarma and Whitelaw focused on the effects of injection angles of coolant streams and lip thickness on the cooling effectiveness in turbine blade film cooling (Sivasgarma & Whitelaw, 1969). This particular study will be looking at the total mass flow rate and the cooling effectiveness. The mass flow rate is the summation of all the coolant channels mass flow rates that are utilized for the desired cooling effects. If the cooling configuration utilizes less total mass flow rate but achieves the desired cooling then the configuration has been improved. A common expression utilized to express cooling effectiveness, ϕ , of a system is shown below (Cunha):

$$\phi = \frac{T_g - T_{metal}}{T_g - T_{Cin}} \quad (4)$$

Where T_g represents the temperature of the gas in the system or fluid domain, T_{metal} is the temperature of the solid, and T_{Cin} is the coolant temperature entering the passages. The cooling effectiveness is highly useful because it quantifies the performance of the cooling circuit inside the turbine airfoil. The cooling effectiveness ranges from zero, non-existent or no cooling effect, to the cooling effect equaling unity, meaning the airfoil metal and coolant temperature are the same (Cunha). Thus outlining the varying types of cooling and the different types of measuring of the coolant as mentioned above. The rest of this introduction will be stepping through the individual areas of focus of this work along

with varying experiments, specific to the concentrated section, carried out by other contributors in the engineering community.

2.1.1. Trailing Edge Cooling Concepts

Through various cooling concepts and techniques it is important to note that each part of the blade or vane has specific cooling configurations due to the function of the area or cooling needed. An example of this is mentioned with showerhead cooling in the turbine blade cooling section. Figure 5 displays the varying cooling schemes utilized for the Energy Efficient Engine (E^3) 1st stage high pressure turbine (HPT) vane. Figure 5 shows a total of eight varying cooling schemes such as leading edge cooling hole, mid-cord holes, and impinged pin-fin slots at the trailing edge of the vanes cooling structure, just to name a few. Halila's experiment of the E^3 was to produce an HPT configuration to achieve high levels of turbine efficiencies through the use of a static and rotating structure

Figure 5: (Halila, 1982) experiment on Energy Efficient Engine (E^3) showing various cooling concepts for the 1st Stage High pressure turbine vane

and an active clearance control system controlling the blade trip shroud clearances (Halila, 1982). As explained by the author this experiment also incorporated state of the art cooling technologies for both the blades and vanes being air-cooled in the system.

With Halila's experiment it was also shown that even with the multiple functioning cooling schemes, utilized to cool a vane or blade, there were still areas for improvement. One particular area is shown in Fig. 6. It can be seen that even with the

Figure 6: (Halila, 1982) experiment on Energy Efficient Engine (E^3) showing the varying temperature for the 1st Stage High pressure turbine vane

advanced cooling techniques, such as impinged pin-fin slots, in the trailing edge region of the vane, the overall temperature is still significantly higher than other areas of the E^3 HPT vane.

Other experiments using trailing edge configurations in the Mark II and C3X turbine vanes have been carried out and discussed in further detail below. One such experiment carried out by Wang displays varying temperature profiles for different optimization cool channel configurations (Wang, 2015). The author displayed the contour

plots of the different cases and the trailing edge region of the turbine blade can be seen to be almost 100-350 K hotter than the coolest part of the vane. The author demonstrated that the cooling configuration of a vane can make a dramatic difference based on the cooling channel location and size. With these results the authors experiments show an optimized cooling configuration obtained a temperature drop of 40-50 K for the whole vane. The particular areas not effected, which can be seen in the results of the authors work, were the suction side near the vane head and the trailing edge region.

Another experiment that focused on the trailing edge cooling of turbine blades is presented by (Yang & Hu, 2012). This particular experiment was done to quantify flow characteristics of wall jets in the trailing edge of the cooling turbine blade. The authors utilized varying measuring techniques such as particle imaging and pressure sensitive paint to display the flow field's evolution of unsteady vortices and turbulent flows. The authors then went on to explore and optimize varying configurations for improved cooling effectiveness, as discussed in the previous section, to provided protection to the critical portions of the turbine blade. The authors stated one of the most difficult regions to cool is the blade's trailing edges, due to geometric constraints in combination with aerodynamic demands. The aerodynamic demands Yang & Hu references are the desired reduction of aerodynamic losses at the trailing edge. To mitigate these losses the trailing edge of vanes and blades are designed to be as thin as possible. This thin design of trailing edges comes with a significant penalty. The penalty of a thin trailing edge is the structural integrity of the blade or vane is compromised due to the heavy geometric restriction and the amount of coolant design requirements. Yang & Hu summarize that is has been found that most catastrophic failures for turbine blades commonly originate at

the edges-trailing edges, tips and roots (Yang & Hu, 2012). These common failure modes are due to melting, cracking, or erosion that greatly reduces the service life time and the maximum allowable TIT.

With the edges being a critical hotspot, literature addressing the thermal issues pertinent to trailing edge cooling of turbine blades and vanes is scarce prior to (Yang & Hu, 2012) experiment. The experiments performed by (Denton, 1993), (Holloway, 2002), (Medic & Durbin, 2005) spanned across various trailing edge turbine blade effects. Denton focused on aerodynamic losses by gases being mixed to the blade surfaces causing overheating (Denton, 1993). Medic & Durbin produced numerical simulations of 3-D unsteadiness occurring in slot jets due to poor protection of the trailing edge (Medic & Durbin, 2005). Medic & Durbin predicted a 3-D vortex shedding effecting the transporting of heat away from the trailing edge region (Medic & Durbin, 2005). Holloway provided experimental data that found the trailing edge region cooling isn't as effective as anticipated due to the hot gases rapidly reaching the surface of the blade or vane (Holloway, 2002).

These experiments have lead up to some conceptual trailing edge blade and vane configurations such as the weave design and the triple impingement concept. Both of these designs were tested computationally and proved to have the desired mitigation of the thermal effects within the trailing edge region of the vane and blade.

The weave design presented by Weaver is a concept design using two layers of 45° angled channels with a square cross section that resulted in flow passages with constant distributed flow direction, a high surface area, and no jet to jet interactions (Weaver, 2015). This specific configuration had a CHT analysis applied comparing the

effects against the triple impingement design for the trailing edge region. Weaver concluded that the weave design outperforms the triple impingement design both thermally and in loss of pressure (Weaver, 2015). Based off of Weaver's weave design data it shows great promise in the trailing edge region of a turbine vane and blade but

Figure 7: Comparative Image of two different trailing edge cooling concepts a.) weave b.) triple impingement (Weaver, 2015)

could present problems by means of manufacturing (Weaver, 2015). The weave concept design can be seen compared to a basic triple impingement configuration in Fig. 7.

The triple impingement configuration seen in Fig. 7 (b) above has been studied through a wide range of CHT experiments, flow analysis, and geometric patterning. These experiments can be seen the works of (Chyu, 2002), (Chen, 2006), and (Lui, 2012). Chyu contributed a study exploring the internal heat transfer in a triple cavity cooling structure with a ribbed lip for a turbine trailing edge (Chyu, 2002). The triple impingement configuration investigated by Chyn can be seen in Fig. 8.

Figure 8: Side view of a three cavity structure in a Turbine trailing edge (Chyu, 2010)

Chen produced a unique design in the trailing edge of the turbine blade or vane constituting of circular pedestal and oblong shaped blocks (Chen, 2006). The author then utilized transient liquid crystal imaging to determine the local heat transfer coefficient.

Finally, Liu took the trailing edge region of a turbine airfoil and conducted an SST turbulence model to study the flow and heat transfer of a triple-impingement cooling configuration (Liu, 2012). The author also performed a study on the considerable temperature variation in the turbine material depending on the location of the jet of impingement. With the trailing edge region of the turbine vane or blade identified as an area of improvement two different types of cooling schemes will be combined to predict the CHT parameters of the combination. These two concepts are optimized near wall cooling passages and a triple-impingement configuration in the vane trailing edge

2.1.2. Jet Impingement Cooling

One type of cooling scheme this particular paper is going to be focusing on is jet impingement cooling, particularly a triple jet impingement cooling configuration implemented into the trailing edge of a turbine blade. As stated by Han, jet impingement cooling has the most significant potential to increase the local heat-transfer coefficient in any environment it is utilized in (Han, 2006). The jet is a high velocity coolant that is ejected by slots or holes that impinges on a heat transfer surface. Han then goes on to state the jet produced in jet impingement cooling develops a stagnation point where a thin high turbulent boundary layer is produced near the impingement region allowing for significant augmented heat transfer properties to be produced (Han, 2006). Figure 9

Figure 9: Flow regions of a single impinging jet a) complete jet structure and typical region development b) is the core development of the jet. (Han, 2013)

shows the flow region of a single jet impinging on a surface and the coolant distribution along the surface for better effectiveness.

With this potential increase in heat transfer there are some known limitations to jet impingement. One limitation is the placement, due to the design; the need of multiple jets

to make the impingent useful is mostly seen in the leading edge and mid span of a blade due to thickness of the material in those areas. Also the increased pressure gradient along the bulk flow that occurs when multiple impingements are aligned can become problematic because the higher pressure created inside the jet, is higher than the outlet pressure. This is a concern because of possible rupture of the material in a thin region such as the trailing edge of a turbine blade or vane (Chyu,2002). However because of the jet impingements proven effectiveness along the leading edge and mid span of a turbine blade it is currently being studied to be utilized in the trailing edge as well. Jet impingement is one idea that has led to a better correlation between the heat transfer properties of the coolant and the turbine blade material, thus extending the life and producing a better efficiency for the turbine system. This has been experimentally shown by researchers such as Zuckerman & Lior, who provided multiple flow and heat transfer phenomena correlations, mathematical models and a quantitative assessment of impinging jet regions based off of varying turbulence model assumptions, jet geometry, and jet placements (Zuckerman & Lior, 2006),.

Martin produced data for numerous studies summarizing various jet impingement configurations along with heat transfer trends for those varying configurations (Martin, 1977). One such work by the author outlines the hydrodynamics of impinging flow taking in the effects of the variable and boundary conditions for heat and mass transfer. The author then goes on to correlate and investigate the varying conditions for both the single impinging jet nozzle and then an array of nozzles. The authors study eventually resulted in optimal spatial arrangements of arrays of jet nozzles with high performance capabilities.

Research by Uysal investigated the effects of impingement heat transfer with jet arrays relating to the “crossflow” or the convection influence of the upstream jet propagating downstream (Uysal, 2006). This was done by the author while varying the size and hole of the jet configuration. Another study performed by Al-aqal dealt with the “crossflow” of jet impingement as well (Al-aqal, 2003). Al-aqal investigated the effect of the size of the jets, number of jets, and jet-to-target plate spacing on the heat transfer distribution of the jet and target plate (Al-aqal, 2003).

Further application was shown by Yamane who presented impingement cooling as being utilized in the cooling of turbine blades for its extremely efficient cooling performance due to its ability to produce high kinetic momentum (Yamane, 2012). This experiment clarified the effects of jet array impingement heat transfer through injection parameters, jet-to-jet spacing, number of jets and injection distance. Figure 10 displays the varying jet nozzle dimensions and a simple array of jets that were utilized for Yamane’s particular experiment.

Figure 10: Square Jet Array utilized by (Yamane, 2012) to clarify Jet Array impingement heat transfer.

Other varying uses for the jet array were also being investigated such as the one Glynn presented (Glynn, 2015). Glynn's study had a particular interest in cooling of electronic components with the ability of dissipating extremely large heat fluxes (Glynn, 2015). The information presented reported the surface heat transfer measurement for a range of test parameters such as Reynolds Number, jet to target spacing, and finally the jet diameter. The author then produced results in the form of local and full-field heat transfer coefficients with the same range conditions.

Finally Metzger studied and measured the two-dimensional arrays of jets impinging on a surface to parallel jet plate (Metzger, 1979). The overall conclusion determined that both the channel height and hole spacing can have significant effects on the stream wise profiles. With all the varying parameters that could affect the heat transfer capabilities using jet impingent it was important to utilize a correct set of parameters for the application of the jet or jet array in the trailing edge of the turbine vane. As discussed earlier due to the thin region of the trailing edge the next major hurdle was implementation of the triple impinging jet into a cooling configuration. Through further research it was seen that various optimization and algorithm techniques were being experimented with in order to produce varying simple cooling channel configurations. This interest has been circulating because of the possible implementation to be able to eliminate weight in the turbine and easier manufacturing procedures of vane and blade configurations.

2.1.3. Micro-Cooling Channels

The common cooling configuration of a turbine blade has a serpentine shaped cooling channel running throughout the center of the blade as seen in Figure 11 below. According to Bunker this specific configurations fundamental aim, in the technology area, is to obtain the highest overall cooling effectiveness with the lowest possible

Figure 11: Industry Standard view of turbine blade/vane Inside Serpentine Shape Cooling Configuration. (Chyu, 2010)

penalty on the thermodynamic cycle (Bunker, 2013). This performance is achieved by a low coolant mass flow rate and cooling effectiveness, known as a high heat transfer coefficient. This particular method has been concentrated on bulk cooling of an individual blade or component. Due to the concentrated effects of bulk cooling, undesirable attributes have arisen in effective cooling of turbines. Some of these effects are listed by Bunker and can be seen below:

- difficulty in distributing cooling where and in what quantity is required
- Non-uniformity of cooling resulting in metal temperature distribution
- Highly complex fluid mechanics leading to heat transfer coefficients that are difficult to measure or predict
- Investment casing complexity/Part-to-part casting variability
- Costly tooling revisions for any cooling changes requiring modification in casing process (Bunker, 2013)

These reasons have altered the focus of the industry and conveyed the importance of researching, development, and redesigning complex cooling channels to new simplistic internal cooling designs. One concept being explored is the concept of micro cooled airfoils (Bunker, 2013). Micro cooling consists of spreading out the cooling network in a series of smaller and highly distributed channels, or sub-channels or elements, providing better uniformity of cooling and lesser in-plane thermal gradients (Bunker, 2013). This particular concept still has many hurdles to overcome in terms of development which is displayed in the varying experiments that have been utilizing cooling channel optimization and even the function of micro-turbines. Several papers outlining optimization of cooling configurations include work produced by (Wang, 2015), (Muller, 2001), (Dennis, 2003), (Sung, 2006), (Bhon, 2003), (Nowak, 2007-2012), along with works of (Verstraete, 2010) just to list a few.

Wang utilized the C3X test, by (Hylton, 1983), to create and study the optimization of various cooling channel optimizations based on a conjugate heat transfer analysis (Wang, 2015). As mentioned by the author numerous studies have been generated for the different problems concerning flow, thermal and structural criteria of convective cooling passages for gas turbine blade and vane optimization. The authors' research optimization uses a genetic algorithm base on a 2-D thermomechanical analysis model based off of a globally convergent method of moving asymptotes. The author explains the convergent

model through the convergence of design variables characterizing geometric shapes of cooling channels, locations, and dimensions, while minimizing the average temperature as much as possible. The author's experiment also focused on utilizing the following RANS turbulence models, listed in Table 1, to produce pressure and temperature data for blade performance characteristics utilized later in the experiments results and conclusions.

Table 1: Turbulence Models Utilized in Wang's Experiment with Characteristics

RANS Turbulence Model	Number of Equations	Characteristics
SST $k-\omega$	2	Pros: blended $k-\omega$ and $k-\epsilon$ to avoid sensitivity to freestream, Adverse pressure gradient, swirling flows no damping function for sublayer resolution Cons: Over predict separation, sensitive to mesh refinement
Standard $k-\omega$	2	Pros: Adverse pressure gradient, swirling flows no damping function for sublayer resolution Cons: Over predict separation, sensitive to inlet B.C.
$k-kl-\omega$	3	Pros: includes effects of transition, laminar, and turbulent flows Cons: rapid transiting flow due to high turbulence, over prediction of heat transfer coefficients (Walters, 2008)
$k-\epsilon$	2	Pros: free-shear layer flows with relatively small pressure gradients Cons: large adverse pressure gradients

Wang then took the 2-D models and extruded them to produce a 3-D model that was utilized in an extensive CHT analysis to find the temperature and stress of the turbine vane by means of the SST $k-\omega$ turbulence model (Wang, 2015). The author then applied the super ellipse as the basic geometry to describe the passage cross section and ran three separate cases; the circular configuration, super ellipse configuration and finally a near-

surface hole configuration with 19 cooling holes 2mm near the vane edge. The results of the research show that optimization of cooling passages effectively enhanced the thermomechanical performance of the redefined C3X cooling channels by reducing the maximum temperature up to 50 K.

Nowak investigated several theories dealing with various parameters that effect the cooling of blade and vane configurations (Nowak, 2007-2012). One specifically is the optimization of cooling configurations based off of an evolutionary algorithm. This particular algorithm imitates real life with the evolution process of living organisms; this method is based on a probabilistic search and an imitation of biological evolution (Nowak, 2007-2012). The author also explored different shapes and concluded that CHT was starting to become a standard. Bhon researched the effects of different hole shapes for the effects of film cooling across the blade (Bhon, 2003).

Verstraete utilized an aerothermal optimization method on the first stage rotor blade of an axial high pressure turbine by CHT and lifetime modeling (Verstraete, 2010). The aerothermal optimization defined the position and diameter of the cooling channels optimized for the maximum lifetime of the blade while limiting the amount of cooling flow.

Muller took an evolutionary algorithm and implemented a realistic automated design cycle of turbine film cooling (Muller, 2001). The algorithm allowed the use of a multi-objective optimization based on the flow parameters surrounding the blade, the cooling hole location, cooling hole size, and heat transfer characteristics of the blade. The author's overall goals were to reduce the coolant mass flow and at the same time achieve a homogenous surface temperature within certain range of temperature distribution. The

author was able to conclude that the automated optimization tool offered a viable alternative to optimization of engineering devices using engineering intuition and

Figure 12: One of (Dennis, 2003)'s optimization cases using PGA optimizer a.) Baseline case b.) PGA optimizer configuration gradient-based methods.

Dennis produced an optimization whose main goal was to minimize the integrated surface heat flux penetrating an airfoil (Dennis, 2003). This had an indirect effect that minimized the amount of coolant need for the removal of heat from the airfoil being cooled. The author had multiple constraints that were implemented in the project such as maximum temperature of any point in the vane was less than the maximum specified value and at the airfoil surface was greater than the minimum specified value. The author went on to accomplish these task by constraining the optimization of locations and radii of a large number of small circular cross-section, straight-through coolant flow, passages of the gas turbine blade being developed. Overall the experiment considered a turbine blade configuration with a maximum of 30 cooling passages seen in Fig.12. This shows one of the optimized cases Dennis produced (Dennis, 2003). It can be seen that the cooling passages in Fig. 12 (b) have smaller diameters in certain areas, such as the group

of passages at the leading edge of the vane, than Fig.12 (a) which was considered the author's baseline case. Convection boundary conditions were approximated for the presence of the external hot gas and internal coolant, and finally 3-D thermoelasticity finite element analysis was utilized for thermal analysis of different vane configurations. In conclusion a 3-D fluid mechanic and a 3-D thermoelasticity finite element analysis would have to be run to provide a full 3-D conjugate analysis.

Sung explored the cooling performance of a hybrid device consisting of a slot jet impinging into a micro- channel utilizing the $k-\epsilon$ turbulent model (Sung, 2006). The author displayed that the vorticity effects greatly influenced the cooling performance outside of the impinging zone along with higher jet Reynolds numbers yielding stronger attachment to the heated surface. Sung then suggested a simplified hybrid cooling geometry to produce lower surface temperatures and smaller temperature gradients across the heated surface (Sung, 2006).

With the varying cooling channel optimizations explained through the experimental results above and the established need for the increase of efficiency of the turbine system, the interactive study of the solid material and fluid must be taken into account. Throughout the literature one consistent type of model has been studied, varied, and focused in on. This is the concept of a conjugate heat transfer analysis.

2.2. Conjugate Heat Transfer

With the known effects of jet impingement cooling and the better heat transfer properties it produces within a system, studying the concepts simultaneously has taken a step forward as well. Conjugate heat transfer, or the study of a process that involves various temperatures within solids, fluids and the thermal interaction of the heating or cooling of the solid object by the flow of air in which it is immersed, has started to be utilized. Modeling of conjugate heat transfer problems in a CFD domain is very difficult and knowing if they are being accurately predicted has been a significant drawback. This is partly due to the solvers' limitations to accurately capture the developing flow and thermal boundary layer acting across the surface of the particular model. The advantage of the conjugate heat transfer study allows an object to be fully immersed and tested in its environment while predicting the properties of the object and the behavior of the flow simultaneously. Numerous studies on heat transfer performance have been carried out on a wide range of cooling configurations but experimental work addressing conjugate heat transfer is limited. One of the most referenced pieces of research utilizing a CHT analysis is the experiment conducted by (Hylton, 1983).

Hylton started carrying out the idea of simple cooling channel configurations and micro cooling in 1983 (Hylton, 1983). NASA tested two turbine blades the Mark II and the C3X turbine vanes, both with ten radial internal cooling passages, producing extensive data of heat transfer results for varying flow conditions. This particular study aimed to produce an experimental database to develop and validate 2-D boundary layer codes able to predict the internal heat transfer coefficient with further extension to film cooling predictions of airfoils. The results displayed and confirmed that when testing

blades and vanes that the overall heat transfer level is strongly dependent on the Reynolds number level. This is based off the average internal heat transfer coefficient being estimated using round channel correlations for the individual cooling channel inlet conditions that were corrected for a thermally developed length. The usefulness of this particular study is very limited due to the simplicity of internal cooling geometry and the limited experimental measurements only taken at the external surface at the airfoil midspan. Hylton's study has been utilized in numerous studies to help benchmark and optimize cooling channels within both blades and vanes (Hylton, 1983).

One study mentioned above was done by Wang utilizing Hylton data to optimize, enhance and demonstrate the varying arrangements of micro-cooling channels (Wang, 2015; Hylton, 1983). Another experiment produced by Mendonça used the measured data of the C3X turbine vane to validate an unstructured finite volume methodology of a combined flow, thermal and stress analysis of turbine blades CFD and CHT solutions (Mendonça). Other varying studies used Hylton for a benchmarking verification of various CFD codes and turbulence models. These particular pieces of work include Facchini who predicted that the realizable $k-\epsilon$ model outperformed the standard $k-\epsilon$ model utilizing STAR-CD (Facchini, 2004). Then there was Luo and Razinsky who used STAR-CD utilizing the V^2f and standard $k-\epsilon$ model (Luo & Razinsky, 2006). Finally another study using different turbulence models to simulate Hylton was performed by York and Leylek who validated commercialized Fluent to evaluate both the standard and realizable $k-\epsilon$ models (York & Leylek, 2003). The overwhelming conclusion made from these different studies displayed is that the V^2f model better predicted Hylton data than both $k-\epsilon$ models.

The particular studies mentioned above all achieved a range of reasonable predicted and measured surface temperatures on the pressure side region of the C3X turbine vane. However the studies did demonstrate a trend of showing significant differences between the measured and predicted temperature for the suction side of the vane. This particular trend showed that the CFD models and codes being utilized were having trouble accurately predicting the varying boundary layer transitions. The various experiments all generated span wise temperature for the airfoil but with extreme differences comparatively to (Hylton, 1983).

Besides the various turbulence models utilizing Hylton data other varying experiments range through countless turbine components and cooling arrangements also employ a conjugate heat transfer analysis. Some of these particular studies are (Coletti, 2012), (Dees, 2010), (Takahashi, 2000 & 2005), (Laskowski, 2008), (Nirmalan and Hylton, 1989) just to mention a few.

Coletti took a cooling scheme made up of a trapezoidal cross-section, one rib-roughened wall, and slots along the opposite walls and studied the conjugate heat transfer in the turbine blade cooling cavity near the trailing edge as seen in Fig.13 below (Coletti, 2012). Figure 13 is a representation of the rig setup the author used to produce CHT results. Coletti imposed a uniform heat flux along the rib-roughened wall and also defined the inlet test section Re to 67,500 (Coletti, 2012). The author then used infrared thermography and finite element analysis to obtain the distributions of non-dimensional temperature and Nusselt number, Nu , pertaining to the rib-roughened wall. Finally the author took the thermal performance through the wall and demonstrated a comparison with convective results and previously published works with the same configuration.

Figure 13: Rig Layout used by (Coletti, 2012) for rib CHT analysis a.) Full rig with highlighted sections b.) Rib setup

Dees's experiment had two major goals it was trying to obtain. First the author wanted to limit the lack of sufficient experimental data of cooling design validation by computational results (Dees, 2010). Dees provided experimental data for a conducting turbine airfoil that surpassed the current benchmark data, taking into account an airfoil that represented an engine components effectiveness levels and common cooling scheme designs (Dees, 2010). The author accomplished this through matching the Biot number of the test vane to the actual engine component. The study had measurements taken of the flow field and the airfoil for varying thermodynamic and aerodynamic properties. The airfoil and several experimental results are seen in fig.14 (a). All these particular properties were measured for both an adiabatic and conducting airfoil with and without film cooling for a variety of internal coolant flow rates and film cooling blowing rates seen in fig. 14 (b) and fig.14 (c) (Dees, 2010). The second goal of the authors was evaluating the convectional gas turbine heat transfer methods. Of particular interest being

Figure 14: Schematic of Airfoil used by (Dees, 2010) and a sample of model results a.) Airfoil b.) Film cooling for conductive airfoil c.) Adiabatic airfoil effectiveness

the adiabatic wall temperature as the driving temperature for heat transfer assumptions.

To accomplish this, the experiments were carried out on both a conducting and adiabatic airfoil while displaying the thermal boundary layer measurements between the two types of airfoils. In conclusion this experiment gave the first experimental evaluation of adiabatic wall temperature as the assumed driving temperature (Dees, 2010).

Some of the remaining experiments that have been produced, but have limiting data to compare against, is work done by Takahashi who focused heavily on a single turbine component (Takahashi, 2000 & 2005). The specific component focused on was the 1st stage HPT blade. The work the author presented was the external surface temperature for both: an internally cooled blade, one with ribbed internal channels and then one with smooth internal channels. Takahashi's work was limiting due to a simple comparison only represented by qualitative thermal paint measurements (Takahashi, 2000 & 2005).

Throughout the literature more studies addressing conjugate heat transfer on internal cooling coupled with film cooling of a gas turbine airfoil is more prevalent than that of

cooling of a component. One of these focused experiments was produced by Laskowski who predicted Hylton's data set computationally (Laskowski, 2008). Looking at the computational data the agreement between the measured and predicted external surface conditions is exceptionally good. However the leading edge region of the vane suction side followed the trend of significantly over predicting the heat transfer coefficient and temperature due to the assumption that the code Laskowski employed had a turbulent boundary condition for the entire vane surface (Laskowski, 2008). Even more studies can be found and are heavily referenced within Dees's work under the literature review (Dees, 2010).

One final study that this paper will be looking into is presented by (Nirmalan & Hylton, 1989). The authors performed experiment matching of Hylton's data by taking the same thermodynamic and aerodynamic measurements to compute the external heat transfer of a turbine vane. The fundamental difference between Nirmalan and Hylton experiments compared to Hylton is the airfoil employed (Nirmalan & Hylton, 1989). (Nirmalan & Hylton, 1989) presented the effects of downstream film cooling with and without leading edge showerhead film cooling consistent with actual engine conditions (Nirmalan & Hylton, 1989). The experiment highlighted that considerable cooling benefits are attainable by the implementation of downstream film cooling. The authors also stated that the process was shown to be a complex interaction of two competing mechanisms, the thermal dilution effect and the turbulence augmentation. This displayed an injection of cold fluid resulting in a decrease of heat transfer and an increase in heat transfer due to the injection process at the same time due to the mechanism. This study by Nirmalan and Hylton ultimately provided the critical experimental data for a turbine

vane, taking into account both leading edge and downstream film cooling, for improved heat transfer design (Nirmalan & Hylton, 1989).

According to Okita & Yamawaki most of the currently available conjugate solvers lack the special thermal modeling for turbomachinery system applications (Okita & Yamawaki, 2002). These particular limitations in conjugate analysis are due to the solvers being limited to an axisymmetric framework, due to turbomachinery non-axisymmetric components which affect the local heat transfer phenomena. The present work takes in account the conjugate heat transfer phenomena through the finite volume CFD solver in Star-CCM+. The finite volume CFD solver allows the simulation of the heat transfer experienced by the gas in the simulation to be taken into account. The purpose of the conjugate heat transfer analysis, according to Montomoli, is to produce reliable and accurate predictions of the temperature field in hot components, ultimately playing a key role in design process of modern gas turbines (Montomoli, 2004). The overall accuracy of the conjugate heat transfer analysis has the capacity to extend the life of critical components through an accurate design of coolant systems allowing for optimization of the components to better mitigate predicted inefficiencies.

The desire to help establish a community environment of studying conjugate heat transfer is the reason this paper will be focusing on utilizing this particular method to study the jet impingement interaction within the micro-cooling channel concept. The current work will focus on mitigating the high temperature profile of the trailing edge region of the turbine vane through implementation of varying cooling configurations.

2.3. Objective Statement

The objective of this research is to serve as an exploratory investigation into the CHT of the coupled system of the optimized micro-cooling channel configuration produced by Wang and also the triple-impingement configuration presented by Liu (Wang, 2015; Liu, 2012). The expectation is to show that both the micro-cooling passage and triple impingement combination will mitigate the high temperature profile of the trailing edge region, produce a better cooling effectiveness, while exploring the flow characteristics and heat transfer of both systems combined. In this thesis, a 3-D model of the C3X turbine vane has been developed and studied using the various configurations listed above to demonstrate that the triple-impingement coupled configuration has the best performance characteristics by meeting the following project milestones:

- Develop and Validate C3X vane model and CHT model
- Develop and Validate C3X vane model and CHT model implementing Micro-cooling concept
- Develop and Validate C3X vane model and CHT model implementing Micro-cooling concept and Triple Impingement concept
- Analysis of CFD test data, mass flow rate of coolant, total pressure change, heat transfer calculations, and CHT data.

3. Methodology

3.1. Introduction

The majority of the research presented in this paper was produced by taking other experimental works and modifying the procedures, test, and analyzing their results to best fit the model being tested. This particular section will be listing out the steps taken, model domain, model set-up, and the implementations during the model set up. There will also be a brief description of critical model differences and sample calculations pertaining to the research.

3.2. Baseline Domain Setup

The C3X blade has a constant cross section, making it easy to model by splines connecting the feature points. This was done in the CATIA v.5 producing an outline. In order to obtain the feature points of the spline design the individual coordinate points were obtained from Hylton's experiment, seen in appendix A (Hylton, 1983). Along with

Hole No.	U--cm (in.)	V--cm (in.)	Diameter--cm (in.)	Cr
1	2.377 (0.936)	1.311 (0.516)	0.630 (0.248)	1.118
2	1.057 (0.416)	1.534 (0.604)	0.630 (0.248)	1.118
3	1.981 (0.780)	3.119 (1.228)	0.630 (0.248)	1.118
4	1.981 (0.780)	4.674 (1.840)	0.630 (0.248)	1.118
5	1.869 (0.736)	6.182 (2.434)	0.630 (0.248)	1.118
6	1.666 (0.656)	7.747 (3.050)	0.630 (0.248)	1.118
7	1.412 (0.556)	9.235 (3.636)	0.630 (0.248)	1.118
8	1.087 (0.428)	10.759 (4.236)	0.310 (0.122)	1.056
9	0.737 (0.290)	12.253 (4.824)	0.310 (0.122)	1.056
10	0.345 (0.136)	13.757 (5.416)	0.198 (0.078)	1.025

Figure 15: C3X finite element grid structure with cooling hole locations shown. (Hylton, 1983)

the cooling hole locations. After the 2D outline was designed a 3D model was extruded to the length of 7.62 cm which is the height of the C3X obtained from Hylton's data (Hylton, 1983). The baseline 3D model was then imported into the software Star-CCM+ in order to run the conjugate analysis for the baseline case and then extended to include the desired micro cooling channels and triple impingement configurations. The cooling channel placement, also presented in Hylton's work, was extruded from the solid domain of the 3D model based on the coordinates presented (Hylton, 1983). Figure 15 shows the coordinates of the C3X blade cooling channel locations along with the mass flow rates for each passage.

Hylton then applied the Cr constant to a Nusselt Number, Nu, expression correction taking into account fully developed thermal boundary layers that account for thermal entrance region effects (Hylton, 1983).

According to Hylton the Cr constant ranged from 1.03 to 1.12 for each cooling channels and is a function of Pr, Re_D , and x/D (Hylton, 1983). This Cr constant was later utilized in this particular study to calculate the Nu and heat transfer coefficient of the ten cooling channels for a comparison between Hylton, Wang, and the current studies baseline configuration heat transfer data (Hylton, 1983; Wang, 2015). Through sensitivity analysis, it was determined that a $1/3^{rd}$, 2.52 cm, 2.52 diameters, of the span of the

Figure 16: 2.52 cm baseline C3X turbine guide vane replication.

computational model was sufficient for simulation to save on computational cost. The reduced design of the baseline C3X blade can be seen in Fig. 16. This modification to the model helped save on computational cost, computational time, and easier modeling of the multiple configurations.

The mass flow rate inlets set for the cooling channels allowed the individual coolant mass flow rate and temperature specifications to be set for each individual channel. The domain inlet was defined as stagnation inlet and the domain outlet was defined as a pressure outlet to capture and model the free stream hot flow. A periodic internal interface was then created between the suction side of the cut domain and pressure side, allowing the model to be simulated as a subsection of a large system with focused results specific to the model. Finally the model also consisted of a solid to fluid interface between the C3X blade, cut domain, and each individual hole. The interfaces were all conformal and helped predict the hot fluid flow properties around the model and also the

Figure 17: Model Setup (Subdomains: hot flow, vane, coolant)

thermodynamic effects the hot fluid flow was having on the model. The layout of the baseline C3X blade can be seen in Fig. 17.

The solid domain or vane, also modeled after Wang's experiment, was modeled to be ASTM 310 stainless steel with a specific heat of $C_{ss} = 502 \text{ J}\cdot\text{K}/\text{kg}$, a density $\rho_{ss} = 8030 \text{ kg}/\text{m}^3$, as well as a temperature varying thermal conductivity displayed below (Wang, 2015):

$$K_{ss} = 0.0155T + 9.9105 \text{ (W} \cdot \text{K}/\text{m}) \quad (5)$$

For the fluid, the hot gas and coolant, Sutherland's formula was used to describe the viscosity, thermal conductivity of the simulation and its properties (Wang, 2015). The specific heat capacity, viscosity, and thermal conductivity are all functions of temperature. The viscosity is represented by μ and the thermal conductivity by K_{fg} and can be seen below:

$$\mu(T) = \mu_0 \left(\frac{T}{T_0} \right)^{3/2} * \frac{T_0 + S}{T + S} \quad (6)$$

$$K_{fg}(T) = \lambda_0 \left(\frac{T}{T_0} \right)^{3/2} * \frac{T_0 + S}{T + S} \quad (7)$$

Where the following constants are defined as $\lambda_0 = 0.0261 \text{ W}/(\text{m}\cdot\text{K})$, $\mu_0 = 1.7894 \times 10^{-5} \text{ Pa}\cdot\text{s}$, $T_0 = 273.11 \text{ K}$, and $S = 110.56$. The Specific heat capacity of the solid, C_{sf} , is represented by the approximated polynomial fitting (Wang, 2015).

$$C_{sf} = a_0 + a_1T + a_2T^2 + a_3T^3 + a_4T^4 \quad (8)$$

Knowing the mass flow rate, $\dot{m}$, cooling channel hydraulic diameter, D_C , coolant viscosity, μ , and the density of coolant, ρ , the hydraulic diameter Reynolds Number, Re_D , was calculated for each cooling channel. The calculation was performed so the outlet pressure matched, forcing the inlet Re_D to match the data in (Hylton, 1983). The Reynolds

number is a good indication on how the flow is acting within each cooling channel, with the current model displaying a turbulent flow regime in the cooling channels. This type of flow is ideal for the maximum amount of heat transfer possible but it creates transitional flow regimes, making it harder for the simulation to produce accurate results. The coolant inlet Re_D number, based off of the hydraulic diameter of each channel, equation used to determine the individual cooling passages is displayed below:

$$Re_D = \frac{\frac{m}{A} D_c}{\mu} \quad (9)$$

The initial reference values for the coolant entering the ten cooling cannels and initial conditions were acquired from Wang's fully developed flow experiment, while the ten cooling channels initial Re_D were acquired from Hylton (Hylton, 1983; Wang, 2015). Both sets of initial data can be seen in the following Table 2.

Table 2: Initial Cooling Channel Input Dimensions and Initial Cut Domain Fluid Flow Conditions

Cooling Channel Input Dimensions					
Channel #	d (mm)	Mass Flow Rate (kg/s)	T _{in} (K)	Re	
1	6.3	0.0222	342	215,420	
2	6.3	0.0221	344	213,850	
3	6.3	0.0218	335	214,370	
4	6.3	0.0218	336	224,400	
5	6.3	0.0225	330	224,170	
6	6.3	0.0225	355	212,300	
7	6.3	0.0216	336	210,020	
8	3.1	0.00744	350	139,180	
9	3.1	0.00477	377	84,320	
10	1.98	0.00256	387	66,940	
Initial Flow Conditions for Domain					
P _i (Pa)	T _i (K)	P ₀ (Pa)	Tu (%)	M ₀	μ _t / μ
413,286	818	254,172	8.3	0.89	30

The percent difference in the inlet Reynolds number for these calculations had a range 0-10% and can be seen in fig. 18 below showing slight differences between the calculated current studies Re_D number and the Hylton documented values (Hylton, 1983). The biggest differences are seen at cooling channels four and ten. It is important to note that

Figure 18: Reynolds Number comparison of Baseline cases for different studies

the current study didn't apply a fully developed flow field assumption to the coolant flow and this is explained in further detail below.

3.2.1. Continua Setup

The next critical step was obtaining mesh independence for the simulation to run the CHT analysis described in both Wang's and Liu's experiments with the SST k- ω turbulence model (Wang, 2015; Liu, 2012). For optimum accuracy several types of meshing parameters were imparted such as the prism layer mesh, polyhedral mesh, and also generalized cylinder mesh. The prism layer mesh is imparted to improve the accuracy of the flow solution model by using the core volume mesh to generate orthogonal prismatic cells next to wall surfaces and boundaries (CD-adapco, 2014). The Polyhedral mesh and the generalized cylinder mesh provide a balanced solution for complex mesh generation problems, while generating an extruded mesh the length of parts considered generalized cylinders. Finally Star-CCM+ has the feature, control volume, that allows the user to coarsen or refine a certain area of the mesh to more specific values. These were implemented in the meshing of the whole model and all of the mesh parameters for the final cell and meshing criteria is listed in Table 3 below.

Table 3: Final Mesh Specifications

Mesh Location							
Hot Gas				Solid		Cooling Channels	
Prism Growth Rate	2.0	Surface Curvature	60	Average Cell Size (cm ²)	0.109	Average Cell Size (cm ²)	0.017
Number of Prism Layers	15	Average cell size (cm ²)	1.206	Max Y+	-	Max Y+	3.38
Max Y+	1.35	Prism Layer Thickness	0.031 cm			# Prism Layers	15

The basic mesh specifications as documented in table 3 above are highlighted in figure 19. Figure 19 (a) shows the surface curvature refinement, fig. 19 (b) shows the number of prism layers around a sharp bend, fig. 19 (c) shows the concept of skewness angle, and finally fig.19 (d) is a representation of a decreasing base size. These mesh specifications are user defined and allows for a better mesh refinement discusses next.

Figure 19: Basic visual Mesh Specifications: a) surface curvature, b) Number of prism layers, c) skewness angle, d) decreasing base size (CD-adapco, 2014)

The different mesh parameters reduce the total cell count, improve the rate of convergence in some cases, and improve the accuracy of the model as a whole (CD-adapco, 2014). The mesh is seen getting finer towards separate surfaces, such as the individual cooling channels and the C3X turbine vane, within the model and then becoming course again towards the fluid domain inlet and outlet. The refinement with the mesh cells consisted of four changes. Mesh base size increase, .001 m to 0.0062 m, surface curvature refinement, 36.0 to 60.0, number of points per circle allowing for more refinement points to be created along curved surfaces, a reduced prism layer growth rate which captures the interface properties and the overall thermodynamic performance

characteristics, and finally a volumetric control region with a tetrahedral/polyhedral density of 2 and cell growth rate of 0.5 and a reduction of relative base size. More specific mesh and physics characterizes applied in the model were a total of 15 prism layers across the vane, a .0062 m base size, which is .086 m relative to the axial cord length of 7.18 cm, a surface growth rate of 1.2, and all Y+ treatment averaging between a range of 1-3.5 to the wall integration. Finally the mesh was refined to 5% of the base size, 0.31 mm, in the vicinity of the vane reducing the overall cell count to 20 M cells for the simulation setup. Overall the different mesh parameters reduce the total cell count, improve the rate of convergence in some cases, and improve the accuracy of the model as a whole. With the following initial conditions and domain setup a mesh independent study was performed on the fluid domain, C3X vane, and cooling channels. Mesh independence was obtained at 24 M cells and can be seen in Fig. 20 which displays a point probe temperature reading at model convergence for the four different mesh counts

Figure 20: Graph Representation of Mesh Independence Study with Percent differences

utilized in the mesh independent study. The mesh consisted of a volumetric control around the vane that produced a 2.9 M cell count for the vane. A cell count of 18.4 M cells was produced for the hot flow domain, leaving the remaining 2.7 million cells distributed across the cooling channels. This mesh also produced a surface max skewness angel of max 89.93° , a surface average volume change of 0.6, and no negative cell volumes existed, thus satisfying the Star-CCM+ recommendations for a quality mesh. The Reynolds numbers, Re_D , produced by the current model were all within a range of 4-16% different then the NASAs Reynolds, Re_D , numbers, μ , and the other varying parameters. All the numbers took in the uncertainty of $\pm 11\%$ for gas temperature presented by NASA, therefore the differences were considered reasonable for the continued exploration of the micro-cooling configuration and the triple impingement configuration. The simulation is displayed in Figure 21 and can be seen with a full mesh

Figure 21: Transparent View of Meshed C3X Baseline Channel Model with Mesh Refinements

and the refinement characteristics described above.

As mentioned above both the SST $k-\omega$ turbulence model and the all y^+ Wall Treatment was applied to the 2nd order discretization density based differential solver. The all- y^+ wall treatment is a hybrid treatment that attempts to emulate the high- y^+ wall treatment for coarse meshes and the low- y^+ wall treatment for fine meshes and is also formulated in order to produce reasonable answers for meshes of intermediate resolutions (CD-adapco, 2014). The primary reason that the y^+ treatment, non-dimensional wall distance, is utilized in conjugate heat transfer analysis is its capability to allow the simulation to integrate and solve for the slope of the temperature of the wall, dt/dy . The wall y^+ treatment is a set of assumptions that capture the effects of wall shear stress, turbulent dissipation and production from the equilibrium turbulent boundary layer theory (CD-adapco, 2014). This is important because of the developing flow characterizes that take place in the fluid region at the very surface, boundary layer, and the direct effects it can have on the heat transfer of the system. Y^+ values ranging between 0-30 everywhere

Figure 22: Histogram displaying y^+ wall treatment per cooling channel

display that the heat transfer characteristics of the fluid flow and solid domain are being integrated correctly and that the convective heat transfer coefficient calculations are accurately displayed. A histogram plot of the all y^+ wall treatment values can be seen for each individual cooling channel and the fluid domain ranging from 0.84-3.38 representing the baseline model in Figure 22. The micro and impingement configuration with in the same range regarding the wall Y^+ values with slight variations within in the ranges. The micro-cooling case had a range of wall Y^+ values of 0.09 -8.12 and the impingement case had wall Y^+ values ranging from .021-7.07.

The Reynolds-averaged Navier–Stokes approximation (RANS) allows the physical turbulent flow to be represented by a time-averaged equation of motion and will be discussed in the next section. When the couple solid energy is activated it applies an integration scheme to an implicit solver which allows the inclusion of secondary gradients to be taken into account at the mesh faces (CD-adapco, 2014). These additional parameters help generate an extremely accurate boundary layer by taking into account finer meshing criteria, approximations about the flow field, and flow interaction for given parts. For faster convergence of the simulation, the model was setup using both a solution driver and convergence accelerator. These drivers normalize and initialize the beginning steps and allow the control of the Courant number. The Courant-Friedrich's-Levy number (CFL) helps control the pseudo time step the simulation utilizes when iterating, a Courant numbers ranging from 1-4, depending on the model, were eventually established to run the simulations to convergence. In order to obtain such a high Courant number without the simulation diverging in the early iterations stages a linear ramping system was utilized. The linear ramp allows the solution to slowly increase the Courant number over

the number of iterations until the simulation reaches a solution or the specified Courant Number.

3.2.2. Turbulence Models

Typical CFD solvers utilize three methodologies to simulate the turbulent flows. The methodologies consist of the Reynolds Average Navier Stokes (RANS) method, Large Eddy Simulation (LES) method, and Direct Numerical Simulation (DNS). RANS is used for most flow simulations because of the robust nature of being able to model all turbulent scales that the simulation could encounter. LES tends to be utilized when large eddies show up in a simulation; the LES methodology resolves the larger turbulence eddies for the simulation modeling the smaller eddies. DNS resolves all scales of turbulence used for very small, low velocity flow and is mostly utilized as a tool to calibrate turbulence models (CD-adapco, 2014). DNS is considered exact but extremely expensive due to the generating of data can be time-consuming (Moin & Mahesh, 1998). The RANS methodology is the industry standard for its flexibility, the RANS equation, and the amount of unknowns that allow it to utilize different flow parameters depicted in the Navier Stokes equations.

The RANS methodology models scale a wide variety of turbulent characteristics and the most common run models where utilized in Wang's experiment. RANS methodology is best studied through instantaneous fluid motion that is both time and space dependent (Sodja,2007). The RANS instantaneous variables consist of a fluids density (ρ), velocity components (u_i), pressure (p), and the components of viscous stress tensor ($T_{ij}^{(v)}$). These variables are then substituted into the momentum equation and the continuity equations where further assumptions, such as incompressible flow or compressible flow, are made

before further modeling is accomplished (Sodja, 2007). RANS methodology also employs the concept of Reynolds averaging which is broken down into three common terms: time averaging, space averaging, and ensemble averaging. Each of these terms is better utilized for certain cases. Time averaging is best considered with stationary turbulence or a non-variance in the flow on the average of time (Wilcox, 1994). Space averaging is appropriate for homogenous turbulence, flow on average that does vary with direction, and ensemble averaging which is considered a general aspect and is both time and space dependent (Sodja, 2007). Reynolds averaging has the benefit of decomposing the flow to both averaged and fluctuating components but also introduces additional variables ($\langle u_{i=1,2,3}^2 \rangle$, $\langle u_1 u_2 \rangle$, etc) which don't have available relationships (Wilcox, 1994). These particular variables leave the solution open which highlights how the RANS models change from flow to flow. In order to solve the turbulence closure problem two methods have been commonly utilized. The first method was introduced by Boussinesq in 1877, who described Reynolds stress in a similar way to the viscous stress, the flow is prescribed another property, turbulent viscosity (Wilcox, 1994). The second was Prandtl's 1925 concept that related the turbulent viscosity to the mixing length through algebraic expression. Prandtl's equation refers to the fact, that no additional transport equations besides the energy, mass and momentum equations are needed (Sadja, 2007). However over the years and various attempts have been made and all have been implemented to further enhance the RANS methodology. Next Kolmogorov, in 1942 developed the $k-\omega$ models, which were the first of its kind, and it related turbulent kinetic energy (k) and the rate of energy dissipation (ω) to the length and time scales of the quantities (Mentor, 1994). Other variations of this approach were utilized to form the model $k-\epsilon$. In 1951

Rotta took the Reynolds stress tensor and successfully modeled it by using a system of partial differential equations (Wilcox, 1994). The particular drawback to this particular approach is the fact it introduces six additional equations describing Reynolds stress and a final equation relating to the turbulence length scale. Mentor in 1994 addressed the problem of sensitivity of the inlet conditions (Mentor, 1994). The author accomplished this by adding an additional non-conservative cross-diffusion term, a blending function, and a modified linear constitutive equation. Finally the other quantities (LES and DNS) resolve the shorter length scales than the RANS models, produce better results, but demand a much greater computing power and time (Apsley, 2004). This particular

Figure 23: Display showing the extent of three methodologies used to solve turbulent flows

concept is seen in Fig. 23 which displays the extent of modeling of the three methodologies used to solve turbulent flows.

The current study utilizes the SST $k-\omega$ turbulence model in order to capture the combined effects of the $k-\omega$ formulation of the boundary layer down to the wall viscous

sub-layer, without any extra damping functions, and a k - ε behavior in the free-stream to define its properties (CD-adapco, 2014). The two basic equations utilized in Star-CCM+ to carry out the computations run in the simulation are as listed below:

$$\begin{aligned} \frac{d}{dt} \int_V \rho k dV + \int_A \rho k (\mathbf{v} - \mathbf{v}_g) \cdot d\mathbf{a} = \\ \int_A (\mu - \sigma_k \mu_t) \nabla k \cdot d\mathbf{a} + \int_V (\gamma_{eff} G_k - \gamma' \rho \beta^* f_{\beta^*} (\omega k - \omega_0 k_0) + S_k) dV \end{aligned} \quad (10)$$

$$\begin{aligned} \frac{d}{dt} \int_V \rho \omega dV + \int_A \rho \omega (\mathbf{v} - \mathbf{v}_g) \cdot d\mathbf{a} = \\ \int_A (\mu - \sigma_k \mu_t) \nabla \omega \cdot d\mathbf{a} + \int_V (G_\omega - \rho \beta f_{\beta^*} (\omega^2 - \omega_0^2) + D_\omega + S_\omega) dV \end{aligned} \quad (11)$$

Where S_k and S_ω are the user-specified source terms, k_0 and ω_0 are the ambient turbulence values in source terms that counteract turbulence decay, the Turbulent Production G_k , the Cross-Derivative D_ω , μ_t the Relation for Turbulent Viscosity, the Turbulent Time Scale, and finally the Model Coefficients β and β^* (CD-adapco, 2014). The varying terms listed above show the complexity of the SST k - ω turbulence model. The SST k - ω effectively tries to account for compressibility, Low-Reynolds number modifications, and Mentors blending function to name a few (CD-adapco, 2014). The effects of the flow that the SST k - ω turbulence model takes into account made it the only turbulence model to be utilized, for the current study.

3.2.3. Conjugate Heat Transfer

Conjugate heat transfer is the ability to analyze a complex system's heat transfer interaction between a solid body and flow, over or inside the body, through reliable and accurate predictions of the temperature field. However accurately modeling both the fluid and solid domains of a simulation is an extremely difficult task due to the multiple interfaces between the hot gas domain, the solid domain, and coolant domain. In order to integrate errors these interfaces all have to have a conformal mesh between each individual interface. In this particular case several different types of flows; turbulent flow and transitioning flow, were present in the simulation. The flow characteristics have to be successfully modeled around and inside the solid to produce accurate heat transfer interactions. These interactions allow the model the flexibility to conduct the conjugate simulation of flow field heat transfer and metal heat conduction. For this to happen there were the Coupled Implicit Energy Solver (CIES) and the SST $k-\omega$ turbulence model have been coupled to investigate the heat transfer in fluid flow and solid domain simultaneously. The Coupled Implicit Energy Solver model is an extension of the Coupled Flow model and together they solve the conservation equations for mass, momentum, and energy simultaneously using a time or pseudo-time marching approach (CD-adapco, 2014). The CIES construction is used primarily to solve compressible flows or flows with domain source terms. The characteristics of the CIES allow for an effective convergence rate ranging from incompressible through supersonic flow regimes independent of the Mach number. Finally the CIES has an enthalpy formulation option that allows for the input choice between the enthalpy and the temperature to be used as the dependent variable when implementing the energy conservation equation, ultimately

affecting the way the energy equation is linearized. The linearization is important because the heat transfer conduction coefficient is directly proportional to the slope at any wall boundary condition.

3.3. Micro-Cooling Case Specifics

With mesh independence obtained, the mesh and physics properties of the domain setup of the original benchmark C3X vane design was redesigned, to replicate Case 3b in Wang's experiment (Wang, 2015). This particular case represents 19 cooling channels, offset 2 mm from the edge of the turbine vane, which can be seen to be equally spaced along the suction and pressure sides of the vane leading to the trailing edge. The last four

Figure 24: a) Baseline Vane configuration b) Micro-cooling configuration (Highlighted Super ellipse dimensions)

cooling channels in Wang's experiment were setup to converge towards the trailing edge as much as possible (Wang, 2015). The length, H , of the 17 super ellipse cooling channel configurations turned out to be 6.5 mm with a width of 2 mm, the two remaining cooling passages were circular, located at the vane head, and had a diameter of 2 mm. Figure 24 (a) shows the baseline vane design and Fig. 24 (b) shows the whole vane with the reversed engineered elliptical and circular micro cooling channel configurations utilized from (Wang, 2015).

Mass flow rates, temperatures, or Re_D inlet conditions were not documented for the micro-cooling case; therefore a series of interpretations had to be made in order to produce results. The Hydraulic diameter, for an ellipse, was calculated using the following equation:

$$D = \frac{4BC(64-16E^3)}{(B+C)(64-3E^4)} \quad (12)$$

Where the constant E was represented by the equation:

$$E = \frac{B-C}{B+C} \quad (13)$$

Once the hydraulic diameter was calculated for every cooling channel the Reynolds number was then calculated utilizing the hydraulic diameter and Eq.(9). The corresponding inferred micro-cooling channel inputs, hydraulic diameter, and Re can be seen in table 4 below. The cut domain inlet flow conditions along with the domain setup stayed the same for the multi-cooling configuration. The Micro-cooling simulations mesh count was 21 M cells with the same meshing, continua, and interface setup as described earlier in the paper.

Table 4: Micro-Cooling Channel Inferred and Calculated Input Dimensions

Micro -Cooling Channel Input Dimensions				
Channel #	d (mm)	Calculated Mass Flow Rate (kg/s)	T_{in} (K)	Re
1	0.0053	0.01881	342	215,420
2	0.0020	.003038	387	86,933
3	0.0053	0.01881	342	215,420
4	0.0053	0.01881	342	215,420
5	0.0053	0.01881	342	215,420
6	0.0053	0.01881	342	215,420
7	0.0053	0.01881	342	215,420
8	0.0053	0.01881	342	215,420
9	0.0053	0.01881	342	215,420
10	0.0053	0.01881	342	215,420
11	0.0053	0.01881	342	215,420
12	0.0053	0.01881	342	215,420
13	0.0053	0.01881	342	215,420
14	0.0053	0.01881	342	215,420
15	0.0053	0.01881	342	215,420
16	0.0053	0.01881	342	215,420
17	0.0053	0.01881	342	215,420
18	0.0053	0.01881	342	215,420
19	0.0020	.003038	387	86,933

Overall there will be some differences between Wang's data and the data collected for this particular experiment because of the interpretation of the mass flow rate and temperature entering the 19 micro-cooling channels. The interpretation was based off of the cooling channel specific location within the vane and then an assumption that with in those areas the coolant entering the cannal would have the same mass flow rate and temperature, for Reynolds number match of the baseline configuration cooling channel inputs. Figure 25 below is the a figure of the baseline vane configuration and the regions the vane was split into in order to interpreted the temperatures and coolant mass flow rate

entering the 19 cooling channels.

Interpretation of the coolant entering individual cooling channels was determined based off of the following criteria, hydraulic diameter of micro-cooling channel; the Re_D numbers were then matched. For Re_D matching the elliptical shaped cooling holes had a D_c of .0053 mm, an assumed T_{in} 342 K, and Re_D number, 21000, matching the inlet cooling characteristics of channel one from the baseline case. For the two circular cooling channels, located at the vane head, with a D_c of .002 mm the cooling characteristics of cooling channel eight, T_{in} of 377 K, and Re_D number of 86000, from the baseline case were utilized for comparison purposes. The two inlet mass flow rates were then back calculated employing Sutherlands law ,Eq. (6), and Re_D , Eq. (9), generating 0.01881 kg/s for the elliptical cooling channels and 0.00308 kg/s for the circular cooling channels. The Reynolds numbers produced by the current model were all within a range of 1- 7.13% of

Figure 25: Baseline Case (a) and micro-cooling case (b) showing regions used to interpret Wang's Experiment temperature and coolant distribution for micro-cooling channels.

the expected Reynolds numbers and was considered reasonable for further implementation into the coupled configuration A representation of the calculated

Reynolds number for both the micro-cooling case and triple impingement case is seen displayed in fig. 26. There are no differences in Re_D entering seen at any of cooling channels between both the triple impingement and Micro-cooling CFD predictions. However there is a highlighted difference between the CFD predicted Re_D and the originally calculated Re_D number calculated for comparison; further exploration on to why the differences between these Re_D has occurred is still needed to be investigated.

Figure 26: Inlet Reynolds Number comparison of C.S. Micro-cooling and Triple Impingent Configuration

3.4. Triple Jet Impingement Combination Specifics

Applying the same meshing, domain setup, and interpretations made for the micro cooling channel experiment, the model was extended to encompass a triple impingement cooling configuration contained within the last three micro cooling channels at the turbine trailing edge. The cooling configuration can be seen in Fig. 27, which highlights the triple impingement offset cooling channels in the trailing edge of the turbine vane. A total of 5 & 6 impingement jets were represented when connecting the cooling channels; these jets were positioned in a staggered formation from one cooling channel to the next.

Figure 27: Isometric Cut View of Dual cooling configuration (Highlighted Trailing Edge Triple Impingement Jet Locations)

Research taken from Glynn concluded that the area averaged heat transfer increases with a decreasing jet diameter and attributed this to the higher jet velocities involved when smaller nozzles were used (Glynn, 2015). This conclusion was referenced during the determination of the impingement jet parameters, position and jet diameter, because

of the micro cooling channel design. Each individual impingement jet has a diameter, D_j , of 0.813 mm so that the ratio of H/D_j is at 7.9 which falls into the injection distance to hole diameter range, 2 to 8, outlined in Yamane's experiment (Yamane, 2012). With the diameter of the impingement jets set in the jet array the distance between each jet had to be determined to produce the most effective heat transfer as described by Yamane as being a jet –to-jet space ratio, S/D_j , of 4 to 8. In order to accomplish this with the diameter of the jet, D_j , 0.813 mm the spacing, S , was determined to be 3.26 mm to have the jet-to-jet spacing ratio to become 4 (Yamane, 2012). Finally it is important to note that Reynolds numbers produced by the current model were all within a range of 1- 16% different than the calculated Reynolds numbers, displayed above, providing confidence that the model was producing accurate results which can be viewed in Fig.23 above.

The jet impingement simulations mesh count was 23 M cells with the same meshing and continua setup as described earlier in the paper. The major difference is the fact that another internal in-place interface had to be utilized in order to collect the flow properties at the trailing edge ejection region of that triple impingement configuration. The internal in-place interface allows for the model to take in account the flow properties happening between the triple impingement cooling channels and the fluid domain flow. The interface is a Fluid-Fluid interface with the added mesh settings of an In-Place interface allowing for the transfer of mass and energy to be taken into account, the continuum quantities such as mass and energy are transferred from one part of the volume mesh to another or allowed for a periodic condition to be applied (CD-adapco, 2014).

Interpretation of the coolant entering the individual cooling channels was determined based Re_D matching. Re_D matching led to some model variations between

the Micro-cooling case and the triple-impingement case. The variation was seen in the individual cooling channel flows. Flows exiting the cooling channels were seen to have choked flow with an exit Mach number of 1 or higher. To keep the flow subsonic and unchoked further matching of the exit Mach numbers of the individual cooling channels was employed between the Micro-cooling configuration and the triple impingement configuration. To accomplish this outlet pressure input had to be varied per cooling channel. This outlet pressure helped balance the air flow in the channel acting as a gauge reducing or increasing the flow speed through the channel. Figure 28 shows a graphical representation of the exit Mach number per cooling channel for both the Micro-cooling configuration and triple impingement configuration. It can be seen that the exit Mach numbers were within a range of 0- 7.33% different between the cases. The major

Figure 28: Exit Mach number comparison of C.S. Micro-cooling and Triple Impingent Configuration

differences seen at the exit Mach numbers are due to the varying Mach numbers the CFD calculated for the cooling channel inlets, and the individual outlet pressure utilized to

make to keep the system subsonic. This had varying effects on the flow within the channel, such as reduced temperature of the flow which affects the energy balances within the channel.

4. Data Reduction

4.1. Introduction

The following section will be comparing Hylton's NASA Run 157, Wang's CFD, and the current studies CFD data for the various cooling configurations. A field mean variance is defined as the average of a field function value as a solution progresses (CD-adapco, 2014), this was implemented in the fluid domain because of the unsteadiness observed. Some simple fluid flow characteristics such as Mach number and pressure will be shown for only the baseline case to exhibit the accuracy of the current model setup and its abilities to capture the C3X vane configurations fluid flow characteristics, other current study flow field contours for the various configurations can be seen in appendix B. Various CFD model versions along with monitoring will also be gone over to highlight the process of setting up the final CFD model. Finally the coordinate system for plots of gauge pressure distribution, temperature distribution, and normalized heat transfer coefficient distribution, h/h_0 , the external airfoil heat transfer coefficient divided by the reference heat transfer coefficient ($1135 \text{ W/m}^2\text{-K}$) is defined (Hylton, 1983).

4.2. Model Variations and Monitoring

This particular section will be reviewing some of the model variations and simplifications made to before the final CFD simulation was run for convergence. The various models explained below helped with the understanding of how the CFD program Star-CCM+ could be varied to obtain a solution. Some of the variations will show important meshing features, interface capabilities, and monitoring effects all used to refine the model for the current study.

One of the varying features presented within the CFD Star-CCM+ software package is the ability to assign a certain region, body, or part to specific interface. The CFD program then allows for the user to utilize a different type of interface to model various real world interactions within the simulation utilizing a less complicated geometry. One such interface that was used successfully within the current study was the periodic internal interface. However the original model presented by Wang stated that the

Figure 29: Graph displaying Monitor plot of activated fully developed interface

coolant channels were considered a fully developed interface (Wang, 2015). While Star-CCM+ has this particular interface that can be applied the user manual indicated a longer time of convergence. To test the convergence, of the summation of the mass flow rate, a simple model consisting of a box fluid domain and a single cooling channel with a fully developed interface was constructed. The simple model was run with the same inlet conditions for both the fluid domain and coolant channel as describe above. The simple model simulation started out without the fully developed interface active and then the

interface was activated at 345 interactions. The major jumps within the graph are due to varying the CFL number of the simple model, speeding up the simulations convergence rate, demonstrated that the convergence rate would have taken double the amount of time to reach a converged solution. This can be seen in Fig. 29 showing the monitor value of the summation of the mass flow rate of the cooling channel, which should be zero, vs the iterations the simple case was tested at. With the verification from the simple model, the decision to run the current study models without the fully developed interface was made to save on computational time.

Another simulation was utilized and the solution displayed residuals that converged based on periodic convergence. This convergence is seen by the slight oscillations of the residuals within a given a range of iterations. With the residual convergence obtained, individual flow parameters were evaluated and it was soon discovered that certain key flow characteristics hadn't converged in the model thus showing the importance of setting up monitors on particular aspects of the model to monitor convergence of a steady state solution. Heavy emphases was put on various aspects of the current study such as: the Mach of the outlet, individual inlet cooling channels velocities, Re_D , the domain inlet and outlet conditions, vane temperature distribution, and the mean average of varying flow parameters. One of these flow parameters can be seen displayed in fig. 30 (a) as the mean average temperature shows convergence yet the maximum temperature of the system hasn't converged to a steady state. Another example of a flow characteristic being monitored is that of the exit Mach number at the outlet of the fluid domain. In this particular case the model didn't show convergence through a constant value or periodic convergence. For this case the outlet

Mach number varied due to pockets of high Mach numbers, low pressure regions, and high negative mass flux flow properties on the prism layers of the mesh in certain regions which is highlighted in fig.30 (b). These particular hot spots were causing reversed flow and supersonic flow characteristics at the outlet of domain and causing non-convergence of a single value, ultimately leading to a diverged model. To fix the problem the models were re-meshed to exclude the prism layers along the fluid domain inlet, outlet, and the periodic interface.

Figure 30: Diverged Mole results for C.S. baseline case highlighting a) Various Temperature monitors and b) high pockets of mass flux

It was also determined that the extruded prism layer feature on the individual

solid to fluid interfaces, between the vane and cooling channels, and the vane to hot fluid flow interfaces was integrated for better flow development and convergence of the solution. This particular feature applies a one to one matching between the prism layer and the cell of the mesh. When this was applied to the model, at the domain edge the one to one matching produced flat meshed cells. These particular cells would create an un-uniformed mesh at the boundary of the fluid domain edge. The cells then would produce extremes in the solution as described above. Through various failed model setups the importance of meshing became apparent and the importance is highlighted as:

The mesh significantly impacts stability, accuracy, and convergence of the simulation along with the success or failure in attaining the computational solution. The mesh has to be sufficiently fine to adequately resolve important flow features and geometry topology, allowing for the capture of flows such as recirculation vortices and steep flow gradients within the viscous boundary layer. CFD solutions thus require proper mesh resolution where these flow features are expected (Pham & Marshall, 2011).

Two specific meshing characteristics that heavily influenced the final model convergence were the implementation of excluding the prism layers along the fluid domain boundaries, as described above, and a control volume, a user defined geometric area specified for any part of the model that allowed for mesh refinement to that particular area only. This helped to produce a finer mesh towards the vane and a course mesh towards the fluid domain edges as seen in Fig. 21 above. This particular feature reduced the cell count of the model by four million cells and allowed for better flow prediction. With the various refined models utilized to find a converged solution certain flow around the vane was seen to be transitioning. The suction side of vane displayed a

higher transitional flow region around the vane head. This particular transition was due to a higher development of a thermal boundary layer in the area and multiple model Mach contour plots display the same concept. Viewing the contour plots it can be seen that an increase in Mach would occur on the suction side of the vane head speeding up the flow. The transitioning flow and thermally developing regions have been a critical point that commercial CFD struggles to predict due to the various types of flow occurring and needing to be accounted for within the model. However the commercial software Star-CCM+ allows for transitioning flow model corrections and calculations that take in account to better predict developing Reynolds numbers and even supersonic flow conditions. These particular extensions have been looked into and should be heavily considered and implanted in future studies to better predict the flow and the conjugate heat transfer characteristics.

4.3. Heat Transfer Reduction

Another highlighted aspect is the calculated average heat transfer coefficient of the individual cooling channels between the data presented by Hylton, Wang, the current study's baseline and a comparison between the micro-cooling configuration and the triple impingement configuration (Hylton, 1983 : Wang, 2015),. The HTC is a quantitative characteristic of the transfer of heat from the metal to the coolant. This can be summarized as the fundamental aim is to obtain the highest cooling effectiveness with the lowest penalty to thermodynamic system (Kirankumar, 2013). For this study the cooling channels HTC should be larger displaying a larger heat transfer, or cooler metal temperatures. Hylton utilized the following equation of Nusselt Number, Nu , to predict the HTC of each cooling channel (Crawford & Kays, 1980):

$$Nu_D = Cr(0.022Pr^{0.5}Re_D^{0.8}) \quad (14)$$

This equation has the correction coefficient Cr , described above, the Prandtl Number, Pr , based off the average coolant temperature, and the Reynolds Number, Re_D , measured from the data extracted by (Hylton,1983). The author then calculated the average heat transfer coefficient from the Nusselt Number, hole diameter, and thermal conductivity.

Wang's study of the individual cooling passages placed a convective boundary condition on his CFD model by assuming fully developed flow (Wang, 2015). This flow took in several different assumptions in order to compute the Nusselt Number (Nu). The Nu was then used to calculate the convective heat transfer coefficient. The author assumed the hot flow around the vane to be constant and uniform. Next the author assumes that the experimental temperature data from Hylton had a fixed thermal boundary condition applied to it. The author then applied a constant thermal load for the heat transfer calculations presented (Wang, 2015). Finally due to the changing parameters of the cooling channels, diameters and Reynolds number, the heat transfer coefficient will differ per cooling channel. To accommodate the varying heat transfer coefficient per cooling channel various aspects of the coolant had to be anticipated. Such as assuming the inlet flow temperature is constant and uniform which is listed by (Wang, 2015).

In order to calculate the convective heat transfer coefficient the Nusslet Number, Nu was defined according to the forced convection Nusselt Number of a tube represented by the Dittus-Boelter equation (Dittus & Boelter, 1930):

$$N_u = 0.023 Re_D^{0.8} Pr^{0.4} \quad (15)$$

Eq. (14) displays the Reynolds number, Re_D , and the Prandlt number, Pr , the Reynolds

number is defined above and the Prandtl number is defined as the following:

$$\text{Pr} = \frac{c_p \mu}{K_{fg}} \quad (16)$$

Eq. (15) expresses the specific heat capacity of the coolant, C_p , the dynamic viscosity, μ , and the thermal conductivity, K_{fg} , both represented by Sutherlands Law shown above. The heat transfer coefficient was then estimated using the Nu and is seen below:

$$h = \frac{K_{fg} \text{Nu}}{D_c} \quad (17)$$

Where K_{fg} is the thermal conductivity and D_c is the hydraulic diameter of the cooling channel. Combining Eq. (9) and Eqs. (15-17) the heat transfer coefficient can be reduced down to Eq. (18) (Wang, 2015):

$$h = 0.02424 \cdot C(\lambda, \dot{m}, \mu) D_c^{-1.8} \quad (18)$$

In Eq. (17) C stands for the coefficient related to a specific cooling channel and is considered constant due to the assumptions made about the coolant above.

$$C(\lambda, \dot{m}, \mu) = \lambda \cdot \left(\frac{\dot{m}}{\mu}\right)^{0.8} \quad (19)$$

The C.S. utilized Newton's law of cooling to compute the HTC. Star-CCM+ calculates the heat transfer coefficient using the following equation (CD-adapco. 2004):

$$h = \frac{\left(\dot{q}''_{\text{conduction}}(a)\right)}{[|a|(T_{ref} - T_f)]} \quad (20)$$

Where T_{ref} , is the specified reference temperature, which was assumed to be the volume average, mean bulk, temperature of the fluid of the Individual channel T_f is the wall boundary temperature, and $\frac{\dot{q}''_{\text{conduction}}(a)}{|a|}$, represents the boundary conduction heat flux of the vector field. The boundary conduction heat flux includes the molecular and turbulent diffusion effects at the boundaries of the fluid region. These calculations were utilized to

help predict the individual cooling channel average heat transfer data. The cooling channel average HTC is gone over in the results section of this report.

4.4. Domain Flow Validation

Figure 31 displays the Mach number, and pressure field means of the flow in the domain. It can be seen that current model renditions show the various flow characters matching for both contour plots. The plots show the corresponding high Mach numbers

coupled with low pressure spots.

For ideal flows with in the turbine the streamline splits at the stagnation point with one leg moving along the pressure side and the other leg moving along the suction side of the blade or vane (Acharya). The pressure gradient from the pressure side to the suction side leads to the development of secondary flow. The lowest pressure occurs on the suction surface located at the passage throat area where the flow velocity is the highest. Conversely the highest pressure is located at the stagnation point located at the leading edge of the blade or vane. From the leading edge to the trailing edge of the pressure side the velocity increases steadily as the pressure incrementally decreases. On the suction surface initially an increase in the velocity is seen but the velocity starts to decline when it encounters the adverse pressure gradients produced by the secondary flows (Acharya). The author then states Due to the adverse pressure gradient there is the potential of boundary layer separation that can form near the suction-side blade surface trailing edge. Another way for boundary layer separation to occur, at the blade trailing edge, is due to a finite trailing-edge thickness which can lead to a distinct wake region. The boundary layer flow separation represents a major source of profile losses in the blade or vane passage. Flow separation is a major issue for blades or vanes with high loading (Acharya). The high loading reduces the suction side pressures allowing for the Mach number and velocity to increase with the local Mach number on the suction side leading to supersonic speeds. As the flow of the blade or vane reaches supersonic speeds local shocks are created and lead to additional aerodynamic losses, such as shock losses or wave drag (Acharya). The author then states downstream of the shock the suction surface pressure rises in the adverse pressure gradient region and boundary layer

separation starts occur earlier leading to increased profile losses for the highly loaded blade or vane.

This developing flow can be seen on the suction side of vane and the stagnation point is clearly seen by the Mach contour as having the lowest speed. The pressure side of the blade shows the expected flow characteristics generally displayed for turbine blades and vanes. The outlet average Mach number the current study output is 0.745 which is 17.7% different than NASAs outlet Mach number of 0.89 (Hylton, 1983). The discrepancies in the exit Mach number calculated by the current model and the data produced by (Hylton, 1983) is best seen with the pressure contour, where the flow shown in the trailing edge region of the outlet domain. The pressure on the lower pressure side shows high and low pressure mixing at the trailing edge of the vane, where some boundary layer separation has occurred and the effects of the secondary flow are being taken into account. This mixing of both pressures varies the Mach numbers, slowing the outlet Mach number of the current study. Overall the flow characteristics produced by the current study match the trends and effects of flow presented from Hylton and Wang (Hylton, 1983 : Wang, 2015). This provided further confidence and verified that the flow of the current study was predicting accurate data.

With model flow trend matching the mid-span data of the whole vane was taken and is defined by the coordinate system seen in Fig. 32 (a). The vane mid-span data was used to exhibition the performance of the different C.S. cooling configurations seen in the results section below. A further extension of the C.S. cooling configurations performance was then focused on the trailing edge of the vane. For the data reduced in the trailing edge section the normalized span coordinate system is seen in Fig. 32 (b).

Figure 32: C3X vane Normalized Span Coordinate Orientations used for data reduction (a) Whole Vane
(b) Trailing Edge

5. Results

5.1. Effects of Cooling

The following section will be comparing Hylton's NASA Run 157, Wang's CFD, and the current studies CFD data for the C.S. various cooling configurations. There will be three individual sections outlining the important features of the particular cooling configuration and data produced and evaluated in the current study. As mentioned above various temperature contour plots, line graphs, and heat transfer calculations will be discussed in further detail.

5.1.1. Baseline Configuration Calibration

Graphical representations of the mean gauge pressure, mean temperature, and mean normalized heat transfer coefficients of Hylton, Wang, and the current models data can be seen Fig. 32 (Hylton, 1983 : Wang, 2015). This data was collected by taking similar span vs axial cord locations (xvs.l), orientation seen in Fig 32.(a) above, to match the Hylton data sensor locations, allowing for the corresponding mean at that location to be displayed (Hylton, 1983). To normalize the data p/p^* , $t/811$ and h/h_0 were calculated using the references taken from Hylton inlet conditions which were conducted at a nominal gas-stream total temperature of 811 K (Hylton, 1983). Figure 33 shows how that the general flow trends acting across the baseline vane matches both sets of data presented by Hylton and Wang (Hylton, 1983 : Wang, 2015). In Fig.33 both the pressure and temperature distributions follow the trend of data presented by Hylton with an under prediction for the pressure occurring at turbine vane head location and a slight over prediction at the vane head and the suction side trailing edge for the temperature. Wang's data can be seen to over predict the trend for the pressure distribution and under predict

Figure 33: Current Study Mid-span data compared to C3X vane data for different flow characteristics.

the temperature distribution but is still considered reasonable for further experimental use (Wang, 2015). The final plot shows the local heat transfer coefficient across the turbine vane mid-span distribution. The pressure side of the vane is seen following the trend set by Hylton while a significant over predicted heat transfer coefficient is predicted by the

current study around the vane head and on a majority of the suction side. Assessing the average h a 26.24 % difference is shown between the two models. The contribution to the differences in all three plots at the vane head is CFDs limited modeling capabilities of capturing transitioning flow and developing thermal boundary layers. Along with the uncertainty documented for the temperature, $\pm 11\%$, by (Hylton, 1983).

Figure 34 is a display of the temperature (K) distribution on the top view of the baseline blade configuration. The top displays both of the models are predicting a similar temperature profile with slight variations. The current model produced a higher temperature profile specifically near the edge regions and is extremely highlighted on the suction side of the blade. Theses discrepancies come from CFDs limiting computations as described above. The middle of the blade seems to be higher in temperature ranging by

about 15-40 K difference. However the temperature contours for both studies have the same general profile and the vane trailing edge represents the hottest temperature of 690+ K.

Figure 34: Top View Comparison of the Baseline Configuration

5.1.2. Micro-Cooling Configuration Calibration

Through the computation of CHT the mid-span mean temperature are compared between Hylton, Wang, and the current studies data for the micro-cooling channel configuration. Figure 35 shows the mean temperature distributions along normalized axial cord, x/l surface. Figure 35 also shows the temperature trend of the current model micro-cooling configuration following the trend of Wang's experiment and is

Figure 35: Micro-Cooling CHT measured values

considerably cooler than the values established by Hylton. Highlighted in the current model is a significantly cooler trailing edge of the vane compared to Wang and Hylton. The graph displays this by following the blue data set as the x/l length starts to reach both -1.2 and 1.2. One contribution of this trend difference is the interpretation discussed earlier for the micro-cooling channels in this study. Another trend displayed is the higher temperature affecting the vane head region as the data set peaks at 0 due to the developing flow profile and thermal boundary layers.

Figure 36 is a display of the temperature (K) distribution on the top and side view of the current study. Compared to Wang's experiment the top view displays a similar

Figure 36: Top and Side View of the Micro- Cooling Configuration

temperature profile with slight variations. The current model vane seems to have a lower temperature towards the pressure side cooling channels, the middle, and a cooler trailing edge of the vane. The middle of vane shows an approximated temperature difference of 20 K than Wang's experiment with the last two cooling channels have a vivid temperature difference of about 140-150 K (Wang, 2015). Another region to note is the upper right of the vane head, the current model presentations 650 K compared to 575 K displayed by Wang, producing a temperature difference of about 125 K in that region (Wang, 2015). However the temperature contours for both vanes progress along a general profile, emphasizing the trailing edge of the vane to be an area of concern for the vane. The side view of the vane shows the difference in model heights and how the temperature

profile propagates towards the trailing edge of the vane. In the bottom of the trailing edge of Wang's model shows the temperature dissipating from a third of the vane height down while the current model seems to display a constant profile (Wang, 2015). As mentioned previously the differences are largely due to the interpretation of the coolant inlet conditions.

5.1.3. Jet Impingement Configuration Comparison

Next Fig. 37 displays the C.S. baseline case, the current study micro-cooling case and the triple impingement configuration for the mean temperature at the vane mid-span. The mean temperature graph displays the micro-cooling case along with the triple impingement configuration to be very effective for the cooling of the vane. The current study displays a cooler region for both the micro-cooling and the triple impingement configuration at the trailing edge of the vane, but an over predicted and hotter vane head than that of Wang (Wang, 2015). Overall the triple impingement configuration is seen to have a lower average temperature distribution than all other cases but specifically compared to Wang especially highlighted at the trailing edge of the vane (Wang, 2015). The average temperature between the triple impingement configuration of the current

Figure 37: Current study CHT measured values for various vane configurations

study and the micro-cooling configuration of Wang shows a 5.5% difference for better cooling performance.

Subsequently, Fig.38 is a display of the temperature (K) distribution on the top of all three configurations. The top can be seen showing the advantages of the micro-cooling and the triple impingement configurations compared to the baseline current study configurations. The three configurations can be seen to have significantly different temperature profile with variations starting with the middle of the vane. The vanes middle

Figure 38: Top Views Comparing the Temperature distribution of the three configurations produced in current study

has a temperature difference of 60 K for both the micro-cooling and the triple impingement configuration compared to the baseline case. Also the last two cooling channels have a dramatic temperature difference of about 130 K compared to the baseline. Another region to note in the current studies baseline case is the vane head. The baseline current model vane head runs around 690 K where the other configurations displaying a temperature of 650 K so there is a temperature difference of about 40 K in the region. The trailing edge is a major section to note where the current model rendition

seems to be fairly constant all the way through the model. Most importantly the temperature distributions show how the coupled configuration of both the micro-cooling and a triple jet impingement configuration show a greatly reduced temperature profile throughout the blade and the trailing edge of the blade, providing the better cooling configurations.

5.1.4. Trailing Edge Focus

This particular section will be focusing on just the trailing edge of the turbine vane utilizing the current studies data and showing the importance and need for a triple impingement configuration for better cooling. This section will be utilizing a third of the normalized span data to highlight the trailing edge of the vane highlighted in Fig. 39. Viewing strictly just the back $1/3^{\text{rd}}$ of the vane, the micro-cooling configuration has a range of 540K- 615K, starting from the last three micro-cooling holes, which is significantly cooler than the baseline case ranging from 635K -690K. The triple impingement configuration shows a range even cooler than the micro-cooling case with a

Figure 39: Highlighted Side Views of the back $1/3^{\text{rd}}$ Temperature contours of the three configurations produced in current study

range from 525K- 595K or 2.8%-3.3% cooler and a 14.7%-18.95% cooler vane than the original NASA C3X vane. The greatest temperature reduction is seen at the tip of the trailing edge as seen in Fig. 39. The very tip of the vane has a temperature of 700 K for the baseline case and the micro-cooling case has a trailing edge tip temperature of 611.76 K. This is a significantly reduced temperature resulting in a 13.5 % cooler than the

baseline. The triple impingement configuration has a registered temperature of 505.88 K which produces a 32.20% cooler trailing edge tip than the baseline configuration and a cooler tip trailing edge region of 18.95 % than the micro-cooling configuration. This is a significant reduction in temperature at the very tip of the trailing edge which is known for deterioration, corrosion, and failure of the vane due to the high temperatures acting across the first stage vane in the turbine. A graphical representation of the back $1/3^{\text{rd}}$, 2.35 radiuses of the normalized span, orientation seen in appendix fig 32 (b), of the trailing

Figure 40: Current study Trailing Edge Temperature Data for various vane configurations

edge of the current studies data was plotted in fig. 40 highlighting the varying temperature distribution across the current studies three configurations. Most importantly the temperature distributions shows the triple jet impingement configuration greatly reduced the temperature profile throughout the vane, thus providing the best cooling configuration out of the three explored.

5.2. Heat Transfer of Cooling Channels

This section will be looking at the calculated compared to the measured average heat transfer coefficient for the individual cooling channels. The calculations were carried out utilizing Eqs. (14, 18, 20) above.

Figure 41 shows the average heat transfer calculated for Hylton, Wang, and the current studies CFD baseline case. It can be seen that the heat transfer coefficient of each cooling channel of the current study has a slight under prediction for the first eight cooling channels but can be seen to get more drastic as the cooling channels approach the trailing edge of the vane. This isn't unexpected due to the fact the heat transfer coefficient varies in terms of hydraulic diameter of the cooling channel. The hydraulic diameter of the cooling channels reduces as much as 50.79%-68.57% as they approach the trailing

Figure 41: Comparison of Heat Transfer Coefficient of Cooling Channels for Baseline data

edge of the vane. With such a high reduction in hydraulic diameter the studies come out to be slightly different with a range of 3.6-16% different between Hylton, the original

C3X computed data, and the current studies baseline case CFD. Compared to (Wang's computed data the current studies CFD baseline case has a range of 2.7- 29% different between the calculated cooling channel heat transfer coefficients. A few major differences are the fully developed flow assumption isn't applied for the current study. The different Reynolds numbers of the individual cooling channel flow, which is reviewed earlier. Finally the assumptions and the uncertainties used when calculating the HTC using Dittus-Boelter's Eq. (15) and Eq. (14) utilized by Hylton. This is important because it highlights the difference in how Hylton and Wang calculated their average heat transfer coefficients, using Nu, compared to the CFD calculations. This comparison quantifies how proportional the boundary heat flux and the driving temperature difference is monitored between the cases.

The average calculated heat transfer coefficient, h , was then determined in the current studies micro-cooling configuration and triple impingement configuration for

Figure 42: Comparison of the average HTC Cooling Channels for C.S. Micro-Cooling and Triple Impingement Configurations

each cooling channel. Figure 42 shows the differences between the current studies Micro-cooling configuration and the Triple-Impingement configuration average heat transfer coefficient for each cooling channel. Figure 42 shows almost no difference between the cooling channels average heat transfer coefficient, h . The percent difference between the two cases ranged from 0– 5% for both the average heat transfer coefficients for the individual cooling channels. The biggest differences are seen at cooling channel 6-8 and 13-17. This is not what is expected for this comparison, the configuration was expected to show cooling channels 10/11/12, the triple impingement section, to have the highest difference. In Fig. 42 below, cooling channels 6-8 and 13-17 all show that there is a variation of heat transfer between the vane and the coolant. This variation of HTC within the channel affects the bulk temperature, heat flux, velocity, and vane temperature of the channel. All of the channels outlined above had the highest variations in outlet Mach number that was described in the methodology section above. The variation in the channel Mach number affected the overall flow effects in the individual channel. This produced larger than expected temperature difference and a lower velocity near the channel walls. The low velocity drives the wall Y^+ and the wall shear stress values that are utilized in the CHT analysis. This can lead to interface matching issues and is a limiting aspect of CHT, affecting the accuracy of CFD analysis. One particular fix is optimization of the outlet pressure specification and tuning it for each channel. This type of optimization controls the flow speed by producing a pressure difference across the channel. In this particular study it was determined to keep the flow within the channels subsonic, $M < 1$. However, in future work this may not be desirable and is something that needs to be taken into account when setting up the flow of the channels.

The triple impingement case was then investigated to understand why the HTC of the triple impingement channels produced a smaller difference than what was expected.

Figure 43 shows the contour of the HTC per cooling channel for the triple impingement

Heat Transfer Coefficient (W/m²-K)

Figure 43: Triple Impingement Channel HTC Contour Plots a) full vane b) highlighted impingement channels with different contour scale

configuration. Figure 43 shows more of what is expected from a triple impingement configuration with almost all the channels showing a constant HTC distributed throughout the channel. Figure 43 (a) does show that while there are some regions of high HTC in the triple impingement region there are also extreme lows, which are highlighted in the figure. Further inspection of the region highlighted showed there is a HTC being measured in that particular area, however it is so small another contour scale had to be used in order to display the effects. This is highlighted in Fig. 43 (b) which shows the different contour in the trailing edge region of the triple impingement configuration. The particular region highlights a spike of the HTC at the entrance of the jets due to the entrance effects, but no clear stagnation effects that were expected. It can be seen in Fig. 44 that the low regions of HTC, fig. 43, have the highest temperature

Figure 44: Triple Impingement Channel Temperature Contour Plot

concentrations. This temperature difference and the low HTC within the region warrants further investigation of the effects of the flow in the configuration.

In conclusion the trends for both average heat transfer coefficients follow the same general trend producing a 4.65 % better averaged heat transfer coefficient for the triple-impingement configuration than the micro-cooling configuration. However the overall percent difference of the average of the heat transfer coefficient was only 1.16%. This difference warranted an investigation into the primary driver of the triple impingement configurations poor performance.

5.2.1. Triple Impingement Performance Investigation

This investigation is going to discuss the reasons of the unexpected HTC results listed above. Figure 45 is a view of all the cooling channels within the triple impingement configuration highlighting the velocity entering and exiting the individual channels. The thing to notice in Fig. 45 is highlighted in the circle region. This region shows high velocities, 336.63 m/s, exiting the jets. This particular averaged flow is approaching the choked flow condition, .973 M.

Figure 45: Side View of Vane Showing Cooling Channel Velocity Vectors

Figure 45 suggest that the velocity exiting at the trailing edge ejection jet is very low compared to the main channel flow. This suggests that the flow is bypassing the triple impingement channels. This flow behavior is highlighted with the particular recirculation zone which is highlighted in Fig. 46. Figure 46 (a) is representation of the velocity

vectors colored by temperature displayed in the triple configuration. While Figure 46 (b) is a focused view of the velocity vectors being displayed between the 1st and 2nd triple impingement channel. Figure 46 (a) and (b) confirms that the flow is acting as suggested in Fig. 45. Figures 46 (a) and (b) also suggest two effects: one that a restriction on the microchannel flow needs to be explored and two simply employing the triple impingement configuration onto a given channel does not work. The flow restriction

Figure 46: Temperature Vectors of Triple Impingement a) full configuration b) highlighted vectors between 1st and 2nd triple impingement channel

should produce the desired effect of forcing more flow out the trailing edge

The recirculation zone could be caused by a number of issues within the system. One issue may arise from the assumption of Reynolds number match discussed above in the Micro-Cooling Case specifics section. Setting the boundary conditions within the CFD solver has the ability to restrict the model and produce undesirable effects.

Another main issue of a recirculation zone seen in fig. 47 is the flow in the jets triple impingement channels. The flow is leaving extremely slow or skipping the jet completely. This shows that implementing the triple impingement is a balance of momentum and a flow network problem. This flow is represented in Fig. 47. Figure 47 is

Figure 47: Section view of Triple Impingement Jet velocity Magnitude (Highlighted Trailing Edge Ejection)

a section view displaying the jets velocity magnitude of the triple impingement configuration. The two major areas to view are the magnitude between the 1st and 2nd triple impingement channel and the flow ejection at the trailing edge. The flow between the 1st and 2nd triple impingement channel shows almost no movement. While

the trailing edge ejection boxed in the fig.47, has a higher velocity towards the top of the jet and a very low or no flow ejection towards the bottom. In summary flow take the easiest path of resistance: meaning the bulk of the flow will go through the smooth micro channels, and only a little flow trickles out of the triple impingement.

The trailing edge ejection jets mentioned above highlights that there is flow entering the jets at the ejection region. This is seen in Fig. 46 with the temperature difference shown at the ejection region. This higher temperature is due to hot air being driven into the trailing edge jets, which is a result the sum of low mass flow rate ,0.0033 kg/s leaving

Figure 48: Triple Impingement Ejection Jet with recirculation zone

at the interface and a change in temperature of 420 K. Figure 48 show the velocity vectors acting at one ejection jet. The reason the hot air is entering the jets is made by the recirculation zone right at the trailing edge of the vane, marked by the star in fig. 48. The zone sits towards the bottom of the ejection jet forcing the hot flow into the jet creating the undesired effects of cavities. These recirculation zones act like a pressure balance within the system and slow the flow and by adding extra mass flow into the triple

impingement system. This untimely affects the momentum, velocity, and Mach numbers experienced within in the channels.

To alleviate this particular problem implementing a higher backpressure, larger pressure difference, to the cooling channel outlets was implemented. The effects of this

Figure 49: Section view of Triple Impingement Jet velocity Magnitude with larger pressure difference (Highlighted Trailing Edge Ejection)

implementation are seen in Fig. 49.

Figure 49 shows some distinct differences than what is displayed in fig. 49 above. One is the cooling channel magnitude has been slowed considerably and the effects of the jets are seen developing in the third cooling channel. The next major difference seen in Fig. 49 compared to Fig. 47 is the jets at the trailing edge. The boxed region displays a faster profile with less back flow entering the ejection jets and distinct jet flow separation.

A few other key flow features to point out can be seen in Fig. 50 which shows the temperature vectors acting within the system. Looking at the vectors between the 1st and 2nd impingement channels there still seems to be a recirculation zone that has formed.

Figure 50: Temperature Vectors of Triple Impingement full configuration with larger pressure difference

However the overall temperature of the system seems to be hotter, which is shown by the ejection region profile and also the temperatures displayed in the cooling channels. This temperature difference is important because from Newton's Law of cooling a larger amount of heat transfer has the ability to produce larger temperature differences in a system. The higher backpressure of the coolant channels creates a higher pressure difference in the system. This pressure difference acts as a driving force and effectively forces the flow out of the ejection jets into the domain flow. However, even with an increase backpressure the impingement region still needs refinement. The investigation concludes; implementing a triple impingement configuration at the same boundary conditions, outlined above, isn't enough for an improved configuration. Additional work has to be implemented to further the validation of the triple impingement performance.

5.3. Cooling Performance

This section will be looking at several cooling parameters to further explore the performance of the triple impingement case. The cooling performance of the vane highlights differences between the minimum and maximum thermal gradient, total coolant flow rates, and finally the channel inlet surface areas. Table 5 shows the thermal gradient, the sum of channel inlet surface areas and coolant mass flow rate differences based on the three cooling configurations being explored in this paper.

Table 5: Calculated Cooling Performance Parameters

Cooling Performance Parameters				
Configuration	Min	Max	Total coolant mass flow rate $\left(\frac{\text{kg}}{\text{s}}\right)$	Sum of Channel Inlet Surface Areas (m ²)
	$\frac{dT}{dx}$ of solid $\left(\frac{\text{K}}{\text{m}}\right)$			
Baseline	37.91	69305	0.1702	0.002304
Micro-Cooling	99.08	87456	0.3258	0.002324
Jet Impingement	45.40	140025	0.3227	0.002324

The magnitude of both the minimum and maximum temperature gradient of the solid vane is displayed in the table 5 above and this is important because the temperature gradient is an indication of the varying thermal stresses affecting the vane. The indication of the stress is seen by having a higher temperature difference occurring within the vane, so a higher temperature gradient would produce a higher stress within the solid. The temperature gradient correlates to the thermal stress induced which can be seen in the following equation:

$$\sigma = E\varepsilon \rightarrow \sigma = E\alpha(dt) \quad (20)$$

Where σ , is the stress induced by temperature expansion, E is the Young's Modulus, ϵ is the strain. For the second equation α , is the temperature expansion coefficient and dt is the temperature difference. This correlation has the ability to display critical stress points that related to the major differences in the thermal gradient. Table 5 above indicates that the triple impingement case has the highest thermal gradient, 140025, which is a 67.5 % increase from the current study baseline case maximum thermal gradient of 69305, however it had only a minimum of was 45.50 which resulted in a 17.98% increase for the vane compared to the minimum temperature gradient for the baseline case which was measured at 37.91 .This big difference depends on the location of the max temperature gradient within the solid. The temperature gradient could vary closer to the vane head in the triple impingement configuration but not necessarily in the trailing edge of the vane. The next step would to run Star-ccm+'s FE volume solid stress solver (CD-adapco, 2014) to better predict the thermal stresses acting within the vane and is an extension of the current study.

The sum of the total surface area for each channel inlet was then calculated. This was done to show the difference between the baseline case and the micro-cooling configuration. This is important because greater difference in the surface area correlates to a reduced amount of metal in the vane. This can be good, due to a weight perspective, but also detrimental to the vane due to the increased stresses acting within the vane as described in the paragraph above and summarized by Wang. It can be seen that both the impingement case and the micro cooling case show zero difference between the two configurations. This is expected because the same scheme of cooling channel inlets was implemented across the two designs. The slight difference is between the baseline and the

micro-cooling case. As seen in Table 5 there is a 9.35% difference between the cooling channels total inlet surface areas. This correlates to a 9.35% reduction in total metal within the vane configuration. This makes the vane lighter in overall weight, however as discussed above there are higher thermal stress concentrations within the micro and impingement cases. With less material and higher stress concentrations this has potential to cause more failure within the vane structure or lower cycle life. A better indication of how the metal would behave is running the Star-ccm+'s FE volume solid stress solver as described above.

Next the total coolant mass flow rate of all the cooling channels is the summation of the mass flow rate at the outlets of each cooling channel. Table 5 highlights that both the micro-cooling configuration and the triple impingement configuration have almost two times as much coolant used than the Current Study (C.S.) baseline configuration. This is due to the fact that both the current studies micro-cooling configuration and triple impingement configuration have nine extra cooling channels within the configuration that are being taken into consideration. These nine extra cooling channels produce an increase of 62.72% difference in the amount of coolant between the C.S. baseline configuration versus the C.S. micro-cooling configuration. For the triple impingement and increase of only 61.86 % in coolant is needed compared to the C.S. baseline configuration. While it shows a 0.96 % reduction of the amount of cooling required compared to the C.S. micro-cooling configuration. Considering that both the C.S. micro-cooling configuration and triple impingement configurations have twice as many cooling channels as the C.S. baseline configuration and there is a 62% increase in the amount of coolant needed for the vane, the 1% reduction needed by the triple impingement compared to the micro-

cooling configuration. However, utilizing more coolant mass flow rate, this particular parameter isn't a fair comparison to defiantly say the triple impingement configuration is better. In order to produce a better performance comparison, the mass flow rate for all three configurations needs to be constant. This particular correction should be taken into consideration for future studies. There are still many ways, that need explored, to help mitigate the extreme jump in the amount of coolant.

5.3.1. Cooling Effectiveness

A brief study of the cooling effectiveness between the three configurations of the current study was also documented. This study took into account the whole vane cooling effectiveness and just the Trailing edge, 2.52 radiuses of the normalized span, cooling effectiveness.

The specifics can be seen in table 6, which display both cooling effectiveness, ϕ , shown by Eq. (4) above. Both cooling effectiveness were used for a comparison to see if the triple impingement further improves the performance of the vane with minimal losses to the turbine section as a whole. In order to calculate the cooling effectiveness several assumptions about the temperatures had to be made. The coolant temperature, $T_{c_{in}}$, was taken to be an average between all the cooling channels inlet temperatures within the configuration to obtain a general effectiveness of the coolant as a whole. The metal temperature was taken as an average temperature of the solid. Finally the temperature of the gas, T_g , was assumed to be the temperature of the fluid domain.

Table 6 shows that there is a major impact, from the baseline case, on the whole vane cooling effectiveness and is

Table 6: Calculated Cooling Effectiveness

Cooling Effectiveness		
Configuration	Whole Vane	Trailing Edge
	ϕ	ϕ
Baseline	0.5296	0.5906
Micro-Cooling	0.7365	0.7672
Jet Impingement	0.7534	0.8176

compared to the triple impingement case is 34.8%. This is a 34.8% increase

in the effectiveness of the cooling or how well the cooling circuit is performing. However table 6 also highlights only a 2.25% better cooling effectiveness between the Micro-cooling case and the triple- impingement. This is an unexpected difference between the cases because the literature suggests a significant improvement with a triple-impingement configuration implemented.

The unexpected difference displayed the importance to measure the trailing edge, back $1/3^{\text{rd}}$, cooling effectiveness. This trailing edge cooling effectiveness was calculated the same way but only taking into account the triple-impingement channels temperatures.

The channels specific to this study are highlighted in Fig. 51 and can be seen highlighting the trailing edge of the three vane

Figure 51: Trailing Edge Vane channel location view a) baseline b) Micro-cooling configuration and c) Triple- impingement configuration

models. In table 6 it can be seen that triple

impingement configuration produces the highest cooling effectiveness in the trailing edge. Compared to the C.S. baseline case it shows a 32.2% improvement, while the Micro-cooling case shows a 26.01 % better cooling effectiveness. Both show a significant cooling effectiveness just on implementation alone. However table 6 also highlights only a 6.34 % difference between the C.S. micro-cooling configuration and triple impingement configuration. This difference led greater investigation into the primary driver of the triple impingement configurations poor performance within the trailing edge.

6. Conclusion

The current studies primary focus was to implement and establish experimental cooling configuration and perform a conjugate heat transfer analysis in a turbine vane to increase the efficiency of the turbine stage. The increased efficiency across the turbine vane directly relates to a boost in overall efficiency the compressor section sees and ultimate leads to mitigation of compressor performance inefficiencies, directly affecting the shaft power output.

In order to achieve increased efficiency for the turbine over the years the industry has implemented numerous cooling techniques. This particular study focused on the concept of an optimized micro-cooling configuration produced by Wang and triple jet impingement concept to optimize the trailing edge of the turbine vane making it cooler (Wang, 2015). This new configuration was utilizing both configurations for overall better heat transfer characteristics demonstrating that a simplistic cooling configuration is beneficial for implantation in turbine vane cooling.

The current study matched the flow characteristics of Hylton and then extended and displayed the triple impingement configuration has the best temperature characteristics out of all three blade configurations. This was especially highlighted in the trailing edge of the vane by the temperature distributions recorded by the current study. The current study showed the micro-cooling configuration has a range of 540K- 615K, starting from the last three micro-cooling holes, significantly cooler than the baseline case ranging from 635K -700K. The triple impingement configuration displayed a range even cooler than the micro-cooling case with a range from 525K- 595K. Taking the average temperature distribution between all three current models the micro-cooling case average

was 16.6 % cooler than the baseline case where the triple impingement configuration was 18.8 % different, cooler, than the baseline. The triple impingement configuration has the capability to reduce the core temperature of a turbine vane up to 18.8 K. This is a slight increase of 2.2% different than the optimized micro-cooling case presented by Wang (Wang, 2015). However the possible 18.8 K that could be obtained is significant boost of turbine efficiency.

In conclusion the triple impingent configuration can produced an average turbine vane temperature of 18.8 % cooler than the predicted values of Hylton but still doesn't produce a significant total pressure drop across the vane. For a significant reduction in cooling of the turbine vane it would make a huge impact if implemented across a single stage. A desirable total pressure drop would still need to be improved for the best enhancement of overall turbine performance. Finally considering the results show an improvement of 2.2% cooler turbine vane compared to Wang's micro-cooling concept, the cost implications for one stage, 100-250 vanes, has the likely hood to produce a tremendous cost savings within an industry utilizing this type of cooling configuration for turbine application. Further work still needs to be conducted to confirm that a combined triple impingement produces better heat transfer properties.

7. Recommendations

Future work should focus on the involvement of the current study implementing various turbulence enhancements for the developing thermal boundary layer and transitional flow within the fluid domain. Stress concentrations applied by the rotating components and the system as a whole showing a more in-depth study of the integrity of the vane structure. Also the applied Finite Element Solver for Solids within CFD to see the true stress concentration locations and predicted effects the cooling channels could have on the integrity of the vane. Variations of the optimized micro-cooling and triple impingement configurations, hole geometries like the study by Bhon, and hole specifications (Bhon, 2003). As well as further cost and overall turbine performance capabilities of a system implementing the various cool configurations should be taken into account. Further room for development also lies in the varying effects of the triple-impingement configuration integrated with other aspects of cooling technologies to see if the performance of the vane could be increased further than it provides to do. Finally, developing an in house model and rig to experimentally determine the effects of the dual combination could be implemented for further validation.

REFERENCES

- Acharya, S., & Mohmood, G. (n.d.). Gas Turbine Handbook [Turbine Blade Aerodynamics]. Louisiana State University, Baton Rouge, LA.
- Al-aqal, O. M. (2003). Heat Transfer Distributions On The Walls Of A Narrow Channel With Jet Impingement And Cross Flow (Unpublished doctoral dissertation). University of Pittsburgh. Retrieved November, 2015, from http://d-scholarship.pitt.edu/8554/1/osama_alaqal_me.pdf
- Bogard, D. D. (n.d.). Gas Turbine Handbook [Airfoil Film Cooling]. University of Austin, Austin.
- Bohn, D., Ren, J., & Kusterer, K. (2003). Conjugate Heat Transfer Analysis for Film Cooling Configurations With Different Hole Geometries. Volume 5: Turbo Expo 2003, Parts A and B.
- Boles. (2013, August). Gas Power Cycle [PDF].
<http://www.saylor.org/site/wp-content/uploads/2013/08/BolesLectureNotesThermodynamicsChapter8.pdf>
- Boyce, M. P. (1982). Gas turbine engineering handbook. Houston, TX: Gulf Pub.
- Boyce, M. P., Meher-Homji, C. B., & Lakshminarasimha, A. N. (1994). Gas Turbine and Combined Cycle Technologies for Power and Efficiency Enhancement in Power Plants. Volume 4: Heat Transfer; Electric Power; Industrial and Cogeneration.
- Boyce, M. P. (November-December 2000). Advanced Cycles for Combined Cycle Power Plants. Russia Gas Turbo Technology Publication.
- Brayton Cycle. (n.d.). Retrieved October 01, 2015, from https://en.wikipedia.org/wiki/Brayton_cycle

- Bunker, R. S. (2013). Gas Turbine Cooling: Moving From Macro to Micro Cooling. Volume 3C: Heat Transfer.
- CD-adapco. (n.d.). Star-CCM (Version Ver. 9.06.009-R8) [Computer software]. Retrieved May, 2014.
- Chen, S. P., Li, P. W., Chyu, M. K., Cunha, F. J., & Abdel-Messeh, W. (2006). Heat Transfer in an Airfoil Trailing Edge Configuration With Shaped Pedestals Mounted Internal Cooling Channel and Pressure Side Cutback. Volume 3: Heat Transfer, Parts A and B.
- Chodkiewicz, R., Kryszynski, J., & Porochnicki, J. (2000). A Recuperated Gas Turbine Incorporating External Heat Sources in the Combined Gas-Steam Cycle. Volume 3: Heat Transfer; Electric Power; Industrial and Cogeneration.
- Chyu, M. K., Uysal, U., & Lee, P. (2002). Convective Heat Transfer in a Triple-Cavity Structure Near Turbine Blade Trailing Edge. Heat Transfer, Volume 4.
- Chyu, M. K. (2010). Recent Advances in Turbine Heat Transfer: With a View of Transition to Coal Gas Based Systems. 2010 14th International Heat Transfer Conference, Volume 8.
- Cohen, H., Rogers, G. F., & Saravanamuttoo, H. I. (1987). Gas turbine theory. Burnt Mill, Harlow, Essex, England: Longman Scientific & Technical.
- Coletti, F., Scialanga, M., & Arts, T. (2012). Experimental Investigation of Conjugate Heat Transfer in a Rib-Roughened Trailing Edge Channel With Crossing Jets. J. Turbomach. Journal of Turbomachinery, 134(4), 041016.
- Crawford, M. E. and Kays W. M. (1980). Convective Heat and Mass Transfer. McGrawHill.

- Cunha, F. J. (n.d.). Gas Turbine Handbook [Heat Transfer Analysis]. Pratt & Whitney United Technologies Corp., Avon, CT.
- Dees, J. E. (2010). Experimental measurements of conjugate heat transfer on a scaled-up gas turbine airfoil with realistic cooling configuration (Unpublished doctoral dissertation). University of Texas at Austin.
- Dennis, B. H., Egorov, I. N., Dulikravich, G. S., & Yoshimura, S. (2003). Optimization of a Large Number of Coolant Passages Located Close to the Surface of a Turbine Blade. Volume 5: Turbo Expo 2003, Parts A and B.
- Denton, J. D. (1993). The 1993 IGTI Scholar Lecture: Loss Mechanisms in Turbomachines. *J. Turbomach. Journal of Turbomachinery*, 115(4), 621.
- Dittus, F. W. and Boelter, L. M. K., 1930, "Heat Transfer in Automobile Radiators of the Tubular Type," *Publications in Engineering*, Vol. 2, p. 443, University of California, Berkeley
- Facchini, B., Magi, A., & Greco, A. S. (2004). Conjugate Heat Transfer Simulation of a Radially Cooled Gas Turbine Vane. Volume 3: Turbo Expo 2004.
- Glynn, C., O'Donovan, T., & Murray, D. (september 2015). Jet Impingement Cooling. *UK Heat Transfer*, 1-9. Retrieved November, 2015, from <http://home.eps.hw.ac.uk/~tso1/Papers/417.pdf>
- Gunston, B. (2001). *Jane's aero-engines*. Coulsdon: Jane's Information Group.
- Halila, E. E., Lenahan, D. T., & Thomas, T. T. (june 1982). Energy Efficient Engine: High Pressure Turbine Test Hardware Detailed Design Report. National Aeronautics and Space Administration. Retrieved from <http://ntrs.nasa.gov/archive/nasa/casi.ntrs.nasa.gov/19850002687.pdf>

- Han, B., & Goldstein, R. J. (2006). Jet-Impingement Heat Transfer in Gas Turbine Systems. *Annals of the New York Academy of Sciences*, 934(1), 147-161.
- Han, J., Datta, S., & Ekkad, S. (2013). *Gas turbine heat transfer and cooling technology*. Boca Raton, FL: CRC Press/Taylor & Francis.
- Hill, P. G., & Peterson, C. R. (1965). *Mechanics and thermodynamics of propulsion*. Reading, MA: Addison-Wesley Pub.
- Hodges, J. (2015). A full coverage film cooling study the effect of an alternating compound angle (Unpublished master's thesis).
- Holloway, D. S., Leylek, J. H., & Buck, F. A. (2002). Pressure-Side Bleed Film Cooling: Part I — Steady Framework for Experimental and Computational Results. Volume 3: Turbo Expo 2002, Parts A and B.
- Hylton, L. D., Milhec, M. S., & Turner, E. R. (1983). Analytical and Experimental Evaluation of the Heat Transfer Distribution Over the Surface of Turbine Vanes. NASA Report No. CR-168015.
- Kirankumar, K. M., & Basawaraj, D. (November 2013). Conjugate Heat Transfer Analysis of Internally Cooled Gas Turbine Blade. *International Journal of Gas Innovative Research in Science, Engineering and Technology*, 2(11), 6559-6566. ISSN: 2319-8753
- Laskowski, G. M., Ledezma, G. A., Tolpadi, A. K., & Ostrowski, M. C. (2008). CFD Simulations and Conjugate Heat Transfer Analysis of a High Pressure Turbine Vane Utilizing Different Cooling Configurations. ISROMAC12-2008-20065
- Liu, J., Weaver, A., Shih, T. I., Klinger, J., Heneveld, B., Ames, R., & Dennis, R. A. (2012). Flow and Heat Transfer in a Triple-Impingement Configuration for

Trailing-Edge Cooling. Volume 4: Heat Transfer, Parts A and B.

Luo, J., & Razinsky, E. H. (2006). Conjugate Heat Transfer Analysis of a Cooled Turbine

Vane Using the V2F Turbulence Model. Volume 3: Heat Transfer, Parts A and B.

Muller, S. D., L., Walther, J. H., & Koumoutsakos, P. D. (2001). Evolution Strategies for

Film Cooling Optimization. *AIAA Journal*, 39(3), 537-539.

Martin, H. (1977). Heat and Mass Transfer between Impinging Gas Jets and Solid

Surfaces. *Advances in Heat Transfer* Volume 13, 1-60.

Medic, G., & Durbin, P. A. (2005). Unsteady Effects on Trailing Edge Cooling. *J. Heat*

Transfer Journal of Heat Transfer, 127(4), 388.

Meher-Homji, C. (1992). Gas Turbine Axial Compressor Fouling: A Unified Treatment

of its Effects, Detection, and Control. *International Journal of Turbo and Jet*

Engines, 9(4).

Meher-Homji, C. (2004). Gas Turbine Axial Compressor Fouling and

Washing. *Proceedings of the Thirty-Third Turbo-machinery Symposium*, 163-191.

Retrieved February 19, 2016, from

<http://turbolab.tamu.edu/proc/turboproc/T33/t33-18.pdf>

Mendonça, F., Clement, J., Palfreyman, D., & Peck, A. (n.d.). VALIDATION OF

UNSTRUCTURED CFD MODELLING APPLIED TO THE C3X TURBINE

INCLUDING CONJUGATE HEAT TRANSFER. 1-12. Retrieved August, 2015,

from http://www.cd-adapco.com/sites/default/files/technical_document/pdf/C3X

[Turbine CHT ETC-8_198-final.pdf](http://www.cd-adapco.com/sites/default/files/technical_document/pdf/C3X)

Metzger, D. E., Florschuetz, L. W., Takeuchi, D. I., Behee, R. D., & Berry, R. A. (1979).

Heat Transfer Characteristics for Inline and Staggered Arrays of Circular Jets

- with Crossflow of Spent Air. *J. Heat Transfer Journal of Heat Transfer*, 101(3), 526.
- Moin, P., & Mahesh, K. (1998). DIRECT NUMERICAL SIMULATION: A Tool in Turbulence Research. *Annu. Rev. Fluid Mech. Annual Review of Fluid Mechanics*, 30(1), 539-578.
- Montomoli, F., Adami, P., Gatta, S. D., & Martelli, F. (2004). Conjugate Heat Transfer Modelling in Film Cooled Blades. Volume 3: Turbo Expo 2004.
- Nirmalan, V., & Hylton, L. D. (1989). An Experimental Study of Turbine Vane Heat Transfer With Leading Edge and Downstream Film Cooling. Volume 4: Heat Transfer; Electric Power; Industrial and Cogeneration.
- Nowak, G., & Wróblewski, W. (2007). Thermo Mechanical Optimization of Cooled Turbine Vane. Volume 4: Turbo Expo 2007, Parts A and B.
- Nowak, G., Wróblewski, W., & Nowak, I. (2011). Convective Cooling Optimization of a Blade for a Supercritical Steam Turbine. Volume 5: Heat Transfer, Parts A and B.
- Nowak, G., & Wróblewski, W. (2011). Optimization of blade cooling system with use of conjugate heat transfer approach. *International Journal of Thermal Sciences*, 50(9), 1770-1781.
- Nowak, G., & Nowak, I. (2012). Shape design of internal cooling passages within a turbine blade. *Engineering Optimization*, 44(4), 449-466.
- Okita, Y., & Yamawaki, S. (2002). Conjugate Heat Transfer Analysis of Turbine Rotor-Stator System. Volume 3: Turbo Expo 2002, Parts A and B.
- Pham, J., & Marshall, D. (2011). Improved Meshing Technique for the Engine Exhaust of an Over-the-Wing Engine and Circulation Control Wing Configuration. 49th

AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition.

Potter, H. J. (November 27, 1972). The Gas Turbine Cycle. ASME Paper Presented at the Gas Turbine Forum Dinner. ASME Annual Meeting, New York, N.Y.

Shyam, V., Ameri, A., Poinsette, P., Thurman, D., Wroblewski, A., & Snyder, C. (June 2015). Application of Pinniped Vibrissae to Aeropropulsion. Volume 2A: Turbomachinery.

Sivasegaram, S., & Whitelaw, J. H. (1969). Film cooling slots: The importance of lip thickness and injection angle. ARCHIVE: Journal of Mechanical Engineering Science 1959-1982 (vols 1-23), 11(1), 22-27.

Sung, M. K., & Mudawar, I. (2006). Experimental and numerical investigation of single-phase heat transfer using a hybrid jet-impingement/micro-channel cooling scheme. International Journal of Heat and Mass Transfer, 49(3-4), 682-694.

Takahashi, T., Watanabe, K., & Takahashi, T. (2000). Thermal Conjugate Analysis of a First Stage Blade in a Gas Turbine. Volume 3: Heat Transfer; Electric Power; Industrial and Cogeneration.

Takahashi, T., Watanabe, K., and Sakai, T. (2005). Conjugate Heat Transfer Analysis of a Rotor Blade with Rib-Roughened Internal Cooling Passages. ASME paper GT2005-68227.

Uysal, U., Li, P., Chyu, M. K., & Cunha, F. J. (2006). Heat Transfer on Internal Surfaces of a Duct Subjected to Impingement of a Jet Array with Varying Jet Hole-Size and Spacing. J. Turbomach. Journal of Turbomachinery, 128(1), 158.

Verstraete, T., Amaral, S., Braembussche, R. V., & Arts, T. (2010). Design and

- Optimization of the Internal Cooling Channels of a High Pressure Turbine Blade—Part II: Optimization. *J. Turbomach. Journal of Turbomachinery*, 132(2), 021014.
- Walters, D. K., & Cokljat, D. (2008). A Three-Equation Eddy-Viscosity Model for Reynolds-Averaged Navier–Stokes Simulations of Transitional Flow. *Journal of Fluids Engineering J. Fluids Eng.*, 130(12), 121401.
- Wang, B., Zhang, W., Xie, G., Xu, Y., & Xiao, M. (2015). Multiconfiguration Shape Optimization of Internal Cooling Systems of a Turbine Guide Vane Based on Thermomechanical and Conjugate Heat Transfer Analysis. *J. Heat Transfer Journal of Heat Transfer*, 137(6), 061004.
- Weaver, A., Liu, J., & Shih, T. I. (2015). A Weave Design for Trailing-Edge Cooling. 53rd AIAA Aerospace Sciences Meeting.
- Weston, K. C. (1992). *Energy conversion*. St. Paul: West Pub.
- Wilcox, D. C. (1994). *Turbulence modelling for CFD*. CA: DCW Industries.
- Yamane, Y., Ichikawa, Y., Yamamoto, M., & Honami, S. (February 2012). Effect of Injection Parameters on Jet Array Impingement Heat Transfer. *International Journal of Gas Turbine, Propulsion and Power Systems*, 4(1), 27-34.
- Yang, Z., & Hu, H. (2012). An experimental investigation on the trailing edge cooling of turbine blades. *Propulsion and Power Research*, 1(1), 36-47.
- York, W. D., & Leylek, J. H. (2003). Three-Dimensional Conjugate Heat Transfer Simulation of an Internally-Cooled Gas Turbine Vane. Volume 5: Turbo Expo 2003, Parts A and B.
- Zuckerman, N., & Lior, N. (2006). *Jet Impingement Heat Transfer: Physics, Correlations,*

and Numerical Modeling. *Advances in Heat Transfer*, 565-631.

APPENDICES

A. C3X Vane Coordinate

This appendix is a Figure showing the C3X Vane coordinates obtained from Hylton's experiment produced at NASA (Hylton, 1983).

Table III.
C3X vane coordinates.

$R_{LE} = 1.168 \text{ cm (0.460 in.)}$ $R_{TE} = 0.173 \text{ cm (0.068 in.)}$

<u>Position number</u>	<u>x--cm (in.)</u>		<u>y--cm (in.)</u>		<u>Position number</u>	<u>x--cm (in.)</u>		<u>y--cm (in.)</u>	
1	0.1097	(0.0432)	11.6546	(4.5885)	40	7.4849	(2.9468)	-0.0617	(-0.0243)
2	0.3894	(0.1533)	12.1890	(4.7988)	41	7.3188	(2.8814)	0.3559	(0.1401)
3	0.7658	(0.3015)	12.6764	(4.9907)	42	7.1483	(2.8143)	0.7737	(0.3046)
4	1.2723	(0.5009)	13.0233	(5.1273)	43	6.9736	(2.7455)	1.1895	(0.4683)
5	1.8743	(0.7379)	13.1376	(5.1723)	44	6.7950	(2.6752)	1.6035	(0.6313)
6	2.4707	(0.9727)	12.9939	(5.1157)	45	6.6116	(2.6030)	2.0155	(0.7935)
7	2.9835	(1.1746)	12.6538	(4.9818)	46	6.4237	(2.5290)	2.4254	(0.9549)
8	3.3985	(1.3380)	12.1976	(4.8022)	47	6.2309	(2.4531)	2.8329	(1.1153)
9	3.7376	(1.4715)	11.6817	(4.5991)	48	6.0328	(2.3751)	3.2380	(1.2748)
10	4.0272	(1.5855)	11.1364	(4.3844)	49	5.8296	(2.2951)	3.6406	(1.4333)
11	4.2885	(1.6884)	10.5766	(4.1640)	50	5.6203	(2.2127)	4.0401	(1.5906)
12	4.5326	(1.7845)	10.0094	(3.9407)	51	5.4051	(2.1280)	4.4364	(1.7466)
13	4.7648	(1.8759)	9.4369	(3.7153)	52	5.1834	(2.0407)	4.8290	(1.9012)
14	4.9870	(1.9634)	8.8605	(3.4884)	53	4.9548	(1.9507)	5.2177	(2.0542)
15	5.2019	(2.0480)	8.2814	(3.2604)	54	4.7191	(1.8579)	5.6020	(2.2055)
16	5.4110	(2.1303)	7.7003	(3.0316)	55	4.4760	(1.7622)	5.9817	(2.3550)
17	5.6157	(2.2109)	7.1176	(2.8022)	56	4.2248	(1.6633)	6.3564	(2.5025)
18	5.8171	(2.2902)	6.5336	(2.5723)	57	3.9654	(1.5612)	6.7249	(2.6476)
19	6.0160	(2.3685)	5.9487	(2.3420)	58	3.6975	(1.4557)	7.0874	(2.7903)
20	6.2126	(2.4459)	5.3632	(2.1115)	59	3.4204	(1.3466)	7.4430	(2.9303)
21	6.4074	(2.5226)	4.7767	(1.8806)	60	3.1339	(1.2338)	7.7909	(3.0673)
22	6.5997	(2.5983)	4.1897	(1.6495)	61	2.8374	(1.1171)	8.1308	(3.2011)
23	6.7894	(2.6730)	3.6015	(1.4179)	62	2.5314	(0.9966)	8.4615	(3.3313)
24	6.9756	(2.7463)	3.0122	(1.1859)	63	2.2149	(0.8720)	8.7826	(3.4577)
25	7.1575	(2.8179)	2.4221	(0.9536)	64	1.8885	(0.7435)	9.0935	(3.5801)
26	7.3335	(2.8872)	1.8301	(0.7205)	65	1.5519	(0.6110)	9.3932	(3.6981)
27	7.5024	(2.9537)	1.2357	(0.4865)	66	1.2052	(0.4745)	9.6815	(3.8116)
28	7.6624	(3.0167)	0.6391	(0.2516)	67	0.8494	(0.3344)	9.9578	(3.9204)
29	7.8115	(3.0754)	0.4115	(0.0162)	68	0.4999	(0.1968)	10.2116	(4.0203)
30	7.8161	(3.0772)	-0.0053	(-0.0021)	69	0.3848	(0.1515)	10.3035	(4.0565)
31	7.8082	(3.0741)	-0.0516	(-0.0203)	70	0.2822	(0.1111)	10.4094	(4.0982)
32	7.7879	(3.0661)	-0.0935	(-0.0368)	71	0.1938	(0.0763)	10.5273	(4.1446)
33	7.7572	(3.0540)	-0.1288	(-0.0507)	72	0.1212	(0.0477)	10.6556	(4.1951)
34	7.7180	(3.0386)	-0.1542	(-0.0607)	73	0.0650	(0.0256)	10.7920	(4.2488)
35	7.6736	(3.0211)	-0.1681	(-0.0662)	74	0.0264	(0.0104)	10.9342	(4.3048)
36	7.6269	(3.0027)	-0.1699	(-0.0669)	75	0.0064	(0.0025)	11.0802	(4.3623)
37	7.5816	(2.9849)	-0.1588	(-0.0625)	76	0.0046	(0.0018)	11.2278	(4.4204)
38	7.5408	(2.9688)	-0.1356	(-0.0534)	77	0.0216	(0.0085)	11.3741	(4.4780)
39	7.5077	(2.9558)	-0.1026	(-0.0404)	78	0.0569	(0.0224)	11.5171	(4.5343)

Figure A.1.: C3X vane coordinates used to produced vane geometry

B. Flow Field Contours

This appendix is divided by the different flow fields for the three current study configurations: Baseline, micro-cooling, triple impingement and will display a mean Mach, mean pressure, a mean velocity magnitude contour and then finally, cascade views of the counter plots.

1. Current Study Baseline Configuration Flow Contours:

Mean of Pressure (Pa)

Mean of Velocity: Magnitude (m/s)

1.1.1. Current Study Baseline Configuration Cascade Flow Contours:

1.2.Current Study Micro-Cooling Configuration Flow Contour

Mean of Mach Number

0.923

0.831

0.738

0.646

0.554

0.462

0.369

0.277

0.185

0.092

0.000

Mean of Pressure (Pa)

413310.

391162.

369014.

346866.

324718.

302570.

280422.

258274.

236126.

213978.

191830.

Mean of Velocity: Magnitude (m/s)

**1.2.1. Current Study Micro-Cooling Configuration
Cascade Flow Contours:**

1.3.Current Study Triple-Impingement Configuration Flow Contours:

Mean of Mach Number

Mean of Pressure (Pa)

Mean of Mach Number

**1.3.1. Current Study Triple-Impingement
Configuration Cascade Flow Contours:**