1-1-2009

A Numerical Study Of Micro-Jet Impingement Cooling Inside A High Pressure Turbine Vane

Marcel L. De Paz
Ryerson University

Follow this and additional works at: <http://digitalcommons.ryerson.ca/dissertations>

Part of the [Aerospace Engineering Commons](#)

Recommended Citation

De Paz, Marcel L., "A Numerical Study Of Micro-Jet Impingement Cooling Inside A High Pressure Turbine Vane" (2009). *Theses and dissertations*. Paper 1143.

This Thesis is brought to you for free and open access by Digital Commons @ Ryerson. It has been accepted for inclusion in Theses and dissertations by an authorized administrator of Digital Commons @ Ryerson. For more information, please contact bcameron@ryerson.ca.

1514581567

A NUMERICAL STUDY OF MICRO-JET IMPINGEMENT COOLING
INSIDE A HIGH PRESSURE TURBINE VANE

By

Marcel E. Leon De Paz
B. Eng Aerospace Engineering
Ryerson University

A thesis
Presented to Ryerson University

In partial fulfillment of the
Requirements for the degree of

Masters of Applied Science
In the program of
Aerospace Engineering

Toronto, Ontario, Canada, 2009

© Marcel E. Leon De Paz 2009

Author's Declaration

I hereby declare that I am the sole author of this thesis or dissertation.

I authorize Ryerson University to lend this thesis or dissertation to other institutions or individuals for the purpose of scholarly research.

I further authorize Ryerson University to reproduce this thesis or dissertation by photocopying or by other means, in total or in part, at the request of other institutions or individuals for the purpose of scholarly research.

Abstract

A NUMERICAL STUDY OF MICRO-JET IMPINGEMENT COOLING INSIDE A HIGH PRESSURE TURBINE VANE

Masters of Applied Science
2009

Marcel E. Leon De Paz
Aerospace Engineering
Ryerson University

A thorough numerical analysis of micro-impingement cooling for application in high pressure turbine vanes is presented. The fundamental flow of an axisymmetric jet is first modeled and studied to ascertain the validity of the results. Subsequent, a fully three dimensional curved vane is modeled with an in-line impinging array of jet diameters 0.5mm. The analysis reveals that spent air collects with increasing streamwise distance from the leading edge, thereby increasing jet exit velocities across the array. For all cases studied, an increase in jet to target spacing increased the overall Reynolds number of the array, but decreased the average heat transfer rate. Micro diameters of 0.25mm were subsequently studied for full vane geometry. For a given mass flow per unit of heated area, the micro-jets considerably increased the average heat transfer by 63%. Similar enhancements were obtained at a fixed pressure drop percentage, and for a desired average heat transfer.

Acknowledgements

The author would like to thank his thesis supervisor, Dr. Bassam Jubran, for his support, guidance, and creative ideas through undergraduate and graduate research. His academic approach and methodical thinking will benefit me in my future academic and professional endeavors.

Completion of this thesis would have not been possible if it were not for the support, patience, and encouragement of the author's parents, Julia Victoria and Jose Alejandro. The author would also like to thank his brothers, Pablo and Stefano, whom offered confidence, and in many cases, a needed greater perspective.

A special thank you goes out to the author's lab mate, Marc J. Ely. Conference trips would have been less adventurous and informative if it were not for his presence and great friendship.

The author would also like to thank Prof.ir. Jos P. van Buijtenen for supervising the research exchange while at the Technical University of Delft in the Netherlands.

Finally, the author would like to thank the Natural Science and Engineering Council of Canada for their support in funding this research. Additionally, a special thanks to the High Performance Computing Virtual Laboratory which provided the significant computational resources for this research.

Table of Contents

Author's Declaration	iii
Abstract.....	v
Acknowledgements.....	vii
Table of Contents	ix
List of Tables	xi
List of Figures	xiii
List of Appendices	xvii
Nomenclature	xix
1.0 Introduction	1
1.1 Impingement Cooling in HPT Vanes.....	2
1.3 Thesis Objective and Strategy.....	5
1.4 Contributions	6
2.0 Literature Review and Flow Physics.....	9
2.1 Single Jet Impingement.....	9
2.2 Inline Impinging Arrays	14
2.3 Numerical Analysis for Impinging Flows	20
2.4 Summary	22
3.0 Numerical Modeling of a Single Axisymmetric Impinging Jet.....	25
3.1 Geometry and Modeling	25
3.1.1 Mesh Creation.....	26
3.1.2 Numerical Setup and Boundary Conditions.....	27
3.2 Mesh Sensitivity	28
3.3 Turbulence Model Selection	30
3.3 Results.....	31
3.3.1 Impingement Flow Field	31
3.3.2 Stagnation Point Heat Transfer.....	35
3.3.3 Local Heat Transfer	37
3.3.4 Average Heat Transfer	43
3.4 Conclusions	45
4.0 HPT Vane – Validation Model	47
4.1 Vane Geometry	47

4.2 Necessity and Strategy for Validation Vane.....	51
4.3 Modeling and Simplification	51
4.3.1 Meshing.....	55
4.3.2 Boundary Conditions and Numerical Setup.....	57
4.4 Mesh Sensitivity	59
4.5 Results & Validation	63
4.5.1 Crossflow to Jet Mass Velocity Ratio	63
4.5.2 Reynolds Number Distribution & Stagnation Point Heat Transfer	65
4.5.3 Streamwise Resolved Heat Transfer	70
4.5.4 Average Heat Transfer	74
4.6 Conclusion.....	76
5.0 HPT Vane – Macro Jets.....	79
5.1 Geometry & Modeling	79
5.2 Boundary Conditions and Numerical Setup.....	83
5.3 Mesh Sensitivity	83
5.4 Results	85
5.4.1 Pressure and Reynolds Number Distribution.....	85
5.4.2 Crossflow.....	90
5.4.3 Local Heat Transfer	92
5.4.4 Effect of Mass Flow Increase	96
5.4.5 Effect of Increasing the Jet to Target Spacing.....	100
5.5 Conclusion.....	106
6.0 HPT Vane – Micro Jets.....	109
6.1 Geometry and Modeling	109
6.2 Results	113
6.2.1 Reynolds Number Distribution and Crossflow.....	113
6.2.2 Local Heat Transfer	117
6.2.3 Streamwise Resolved Heat Transfer and Overall Average Heat Transfer	120
6.3 Conclusion.....	128
7.0 Conclusions and Recommendations for Future Work	131
Appendix	135
References	157

List of Tables

Table 2.1 – Coefficients for equation 2.16, Florschuetz Nu correlation	18
Table 3.1 – List of air properties at 300°K for the single jet study.....	28
Table 3.2 – Average Nusselt numbers for three grids	29
Table 3.3 – Mesh sensitivity analysis summary	29
Table 3.4 – Average Nusselt number and deviations with experimental correlations $H/D=4.5$, $Re=4000$..	31
Table 3.5 – Average Nusselt number deviation with experimental correlations, $H/D=6$	44
Table 4.1 – Geometric parameters for base vane	50
Table 4.2 – Validation model boundary conditions & property values	59
Table 4.3 – Average Heat transfer for three validation grids	60
Table 4.4 – GCI calculation	60
Table 4.5 – Average Nusselt number deviation (%) for all validation cases	76
Table 5.1 – Key parameters for full HPT vane at $G=4.5$, $H/D=1.58$	85
Table 5.2 – Key parameters for HPT Vane $H/D=1.58$ at $G=2.25$ and $G=6$	96
Table 6.1 – Geometrical parameters of HPT vane models	111
Table 6.2 – Key flow parameters for all three HPT vane models of $H/D=1.58$	112
Table 6.3 – Key flow parameters for all three HPT vane models of $H/D=3$	112
Table 6.4 – Average Nusselt number for all HPT vane models at various H/D ratios and G values	122
Table A.1 – Katti and Prabhu [34] Eq. A.3 coefficients	137

List of Figures

Figure 1.0 – TIT trends [2]	1
Figure 1.1 – Impingement cooling of a 1 st stage HPT vane [8]	3
Figure 1.2 – Impingement inserts for 1 st stage Vane, A: Leading edge, B: Aft.....	4
Figure 1.3 – 2 nd stage vane with single impingement insert.....	5
Figure 2.0 – Single impingement flow regions and definitions.....	10
Figure 2.1 - Typical heat transfer distribution under an impinging jet.....	11
Figure 2.2 – Nusselt distribution for a single circular jet with $H/D < 5$	13
Figure 2.3 – Impingement flow inside an HPT vane [31]	15
Figure 2.4 – In-line impinging array representative of a gas turbine vane [28]	15
Figure 3.1 – Axisymmetric impinging jet geometry	25
Figure 3.2 – Singlet jet mesh, $H/D=6$	27
Figure 3.4 – Local Nusselt Distribution, turbulence models and experimental data from [51], $H/D=4.5$, $Re=4000$	30
Figure 3.5 – Streamlines illustrating the toroidal vortex formed at $H/D=1$, and Reynolds numbers 2000-6000.....	32
Figure 3.6 – Streamlines illustrating the toroidal vortex formed for $Re=3000$, and $H/D=1, 2, 3$	33
Figure 3.7 – Velocity contours (m/s) for $H/D=6$, $Re=2000$ and 6000	34
Figure 3.8 – Free jet thickness with increasing Reynolds number, $H/D=6$	34
Figure 3.9 – Stagnation Nusselt number change with Reynolds number and H/D	35
Figure 3.10 – Comparison of predicted stagnation Nusselt values and experimental data.....	37
Figure 3.11 – Local Nusselt distribution with increasing Reynolds number at $H/D=6$ and $H/D=1$	38
Figure 3.12 – Local Nusselt distribution with changing jet to target spacing at $Re=6000$	39
Figure 3.13 – Nusselt number contours on target plate, effect of increasing Re , $H/D=1$	41
Figure 3.14 – Nusselt number contours on target plate, effect of increasing H/D , $Re=3000$	42
Figure 3.15 – Average Nusselt number with increasing Reynolds number, $H/D=6$	43
Figure 3.16 – Average Nusselt number with increasing jet to target spacing, $Re=6000$ and $Re=3000$	45
Figure 4.1 – 2 nd stage HPT vane cross section [8]	48
Figure 4.2 – rene150 airfoil profile for 2 nd stage vane [8].....	48
Figure 4.3 - Completed vane model.....	49
Figure 4.4 – General flow direction through vane	50
Figure 4.5 – plenum chamber for validation vane ($H/D=1.58$)	52
Figure 4.6 – Preliminary validation vane depicting main surfaces and volumes.....	53
Figure 4.7 – Validation model before meshing.....	54
Figure 4.8 – Validation model cross section and inner impingement cavity	54
Figure 4.9 – Validation model $H/d=1.58$, intermediary mesh	56
Figure 4.10 – Validation model $H/D=1.58$, boundary layer mesh	56
Figure 4.11 – Validation model $H/D=1.58$, jet mesh.....	57
Figure 4.12 – Validation model $H/D=1.58$, boundary conditions	57
Figure 4.13 – G_c/G_j ratio for three grids.....	61
Figure 4.14 – Average spanwise heat transfer coefficient for three grids	61
Figure 4.15 – Local heat transfer distribution for all three grids.....	62

Figure 4.16 – Crossflow to jet mass velocity ratio distribution for $H/D=1.58$, $Re_{avg}=5000$	64
Figure 4.17 – Cross flow to jet mass velocity ratio distribution for $H/D=3$, $Re_{avg}=15000$	64
Figure 4.18 – Reynolds number distribution for $H/D=1.58$, 3 at $Re_{Avg}=5K, 15K$	66
Figure 4.19 – Stagnation values across the array for $H/D=1.58$ at $Re_{avg}=5K, 15K$	67
Figure 4.20 – Stagnation values across the array for $H/d=3$ at $Re_{avg}=5K, 15K$	68
Figure 4.21 – Velocity vectors (m/s) for middle jets at rows 2 and 5, $H/D=1.58$, $Re_{Avg}=5000$, RNG	69
Figure 4.22 – Streamwise resolved NU for $H/D=1.58$ at $Re_{Avg}=5000$	70
Figure 4.23 – Nusselt number contours for $H/D=1.58$ at $Re_{avg}=5K$ and $15K$, Kw SST	71
Figure 4.24 – Streamwise resolved Nu for $H/D=1.58$ at $Re_{Avg}=15K$	72
Figure 4.25 – Streamwise resolved Nu for $H/D=3$ at $Re_{Avg}=5000$	73
Figure 4.26 – Streamwise Resolved Nu for $H/D=3$ at $Re_{Avg}=15000$	74
Figure 4.27 – Average Nusselt number for $H/D=1.58$ at various Re_{Avg}	75
Figure 4.28 – Average Nusselt number for $H/D=3$ at various Re_{Avg}	75
Figure 5.1 – Base vane plenum geometry and streamwise jet positions	79
Figure 5.2 – Base vane plenum geometry and spanwise jet locations	80
Figure 5.3 – Base vane before meshing, $H/D=1.58$	81
Figure 5.4 – cross-section comparison for base HPT vane at $H/D=1.58$ and $H/D=3$	82
Figure 5.5 – HPT base vane ($H/D=1.58$) complete mesh	82
Figure 5.6 – HPT base vane ($H/D=1.58$) boundary conditions	83
Figure 5.7 – Pressure drop across array for three different grids	84
Figure 5.8 – Streamwise resolved heat transfer coefficient for three different grids	84
Figure 5.9 – Local heat transfer distribution on pressure surface for three different grids	85
Figure 5.10 – HPT vane absolute pressure contours (Pa), $H/D=1.58$, $D=0.51mm$, $G=4.5\text{ Kg/m}^2s$	86
Figure 5.11 – HPT vane pressure distribution, $H/D=1.58$, $D=0.51mm$, $G=4.5\text{ Kg/m}^2s$	87
Figure 5.12 – HPT vane velocity contours (m/s), $H/D=1.58$, $D=0.51mm$, $G=4.5\text{ Kg/m}^2s$	88
Figure 5.13 – HPT vane Reynolds number distribution, $H/D=1.58$, $D=0.51mm$, $G=4.5\text{ Kg/m}^2s$	89
Figure 5.14 – HPT vane velocity contours (m/s) streamwise and spanwise, $H/D=1.58$, $D=0.51mm$, $G=4.5\text{ Kg/m}^2s$	89
Figure 5.15 – G_c/G_j ratio for full HPT vane and validation vane (Fig 4.16), $H/D=1.58$, $D=0.51mm$, $G=4.5\text{ Kg/m}^2s$	90
Figure 5.16 – HPT vanes suction surface pathlines, $H/D=1.58$, $D=0.51mm$, $G=4.5\text{ kg/m}^2s$	91
Figure 5.17 – HPT vanes pressure surface pathlines, $H/D=1.58$, $D=0.51mm$, $G=4.5\text{ kg/m}^2s$	92
Figure 5.18 – HPT vane Heat Transfer coefficient contours (W/m^2K), $H/D=1.58$, $D=0.51mm$, $G=4.5\text{ Kg/m}^2s$	92
Figure 5.19 – 1D strip shown for local heat transfer extraction on full HPT vane	93
Figure 5.20 – Local Nu distribution on suction surface for HPT vane, $H/D=1.58$, $D=0.51mm$, $G=4.5\text{ Kg/m}^2s$	94
Figure 5.21 – Local Nu distribution on pressure surface for HPT vane, $H/D=1.58$, $D=0.51mm$, $G=4.5\text{ Kg/m}^2s$	95
Figure 5.22 – Reynolds number distribution for HPT Vane $H/D=1.58$, $D=0.51mm$, at $G=2.25, 4.5, 6$	97
Figure 5.23 – Average Reynolds number with increasing G, HPT Vane $H/D=1.58$, $D=0.51mm$	98
Figure 5.24 – Local Nu_l distribution on pressure surface, HPT Vane $H/D=1.58$, $D=0.51mm$, at $G=2.25, 4.5, 6$	98
Figure 5.25 – Local Nu_l distribution on suction surface, HPT Vane $H/D=1.58$, $D=0.51mm$, at $G=2.25, 4.5, 6$	99
Figure 5.26 – Average Nu on HPT Vane $H/D=1.58$, $D=0.51mm$ at various G values	99
Figure 5.27 – Heat transfer coefficient contours (W/m^2K) for HPT vane, $H/D=1.58$, $D=0.51mm$, various G values	100
Figure 5.28 – Effect of H/D ratio on G_c/G_j ratio for HPT vane at $G=6\text{ kg/m}^2s$	101
Figure 5.29 – Average Reynolds number increase with G for HPT vane at $H/D=1.58$ and 3	102

Figure 5.30 – Reynolds number distribution for HPT vane at $H/D=3$ and $H/D=1.58$, $G=6\text{ kg/m}^2\text{s}$	103
Figure 5.31 – Local heat transfer distribution on pressure surface at $H/D=3$ and $H/D=1.58$, $G=6\text{ kg/m}^2\text{s}$	104
Figure 5.32 – Local heat transfer distribution on suction surface at $H/D=3$ and $H/D=1.58$, $G=2.25\text{ kg/m}^2\text{s}$	104
Figure 5.33 – Average heat Transfer for all H/D ratios with increasing G	105
Figure 5.34 – Heat transfer coefficient contours ($\text{W/m}^2\text{K}$) for HPT vane at $G=2.25$, $H/D=1.58$ & 3	106
Figure 6.1 – Model B depiction, $D=0.25\text{mm}$, $H/D=1.58$, spanwise spacing = $8.16D$, $L/D=2.142$	110
Figure 6.2 – Model C depiction, $D=0.25\text{mm}$, $H/D=1.58$, spanwise spacing = $4D$, $L/D=2.142$	110
Figure 6.3 – Reynolds number distribution for HPT Vane models at $G=4.5\text{ kg/m}^2\text{s}$, $H/D=1.58$	113
Figure 6.4 – Reynolds number distribution for HPT Vane models at $G=4.5\text{ kg/m}^2\text{s}$, $H/D=3$	114
Figure 6.5 – G_c/G_j ratio distribution for all HTP Vane models at $G=4.5\text{ kg/m}^2\text{s}$, $H/D=1.58$	115
Figure 6.6 – G_c/G_j ratio distribution for all HPT Vane models at $G=4.5\text{ kg/m}^2\text{s}$, $H/D=3$	115
Figure 6.7 – Pathlines illustrating crossflow pattern for base model A and model C, $H/D=1.58$, $G=4.5\text{ kg/m}^2\text{s}$...	117
Figure 6.8 – Local Nusselt distribution on pressure surface for all three HPT Vane models, $H/D=1.58$, $G=4.5\text{ kg/m}^2\text{s}$..	118
Figure 6.9 – Local Nusselt distribution on suction surface for all three HPT Vane models, $H/D=1.58$, $G=4.5\text{ kg/m}^2\text{s}$...	118
Figure 6.10 – Heat transfer coefficient contours ($\text{W/m}^2\text{K}$) for all HPT vane models at $H/D=1.58$, $G=4.5\text{ kg/m}^2\text{s}$..	120
Figure 6.11 – Streamwise resolved Nu_L for all HPT vanes at $H/D=1.58$, $G=4.5\text{ kg/m}^2\text{s}$	121
Figure 6.12 – Streamwise resolved Nu_L for all HPT vanes at $H/D=3$, $G=4.5\text{ kg/m}^2\text{s}$	121
Figure 6.13 – Average Nusselt number with changing G and H/D , all HPT vane models	123
Figure 6.14 – Pressure drop across HPT vanes with increasing mass flow, $H/D=1.58$	124
Figure 6.15 – Average Nusselt number for HPT vanes with increasing pressure drop, $H/D=1.58$	125
Figure 6.16 – Mass flow per unit of heated area with increasing average Nusselt number, $H/D=1.58$...	127
Figure A1 – Validation vane $H/D=1.58$ illustrating cross-flow mid planes.....	135
Figure A2 – Effective average area for each jet – validation model	138
Figure A3 – Segmented Airfoil Surface, validation model, $H/D=1.58$	139
Figure A4 – HPT base vane mesh, $H/D=1.58$, $D=0.51\text{mm}$	140
Figure A5- HPT base vane mesh, boundary layer, $H/D=1.58$, $D=0.51\text{mm}$	141
Figure A6 – Crossflow mid planes for G_c/G_j extraction, full HPT vane, $H/D=1.58$, $D=0.51\text{mm}$	142
Figure A7 – Local heat transfer distribution on pressure surface for two grids, model B, $H/D=1.58$, $D=0.25\text{mm}$..	143
Figure A8 - Local heat transfer distribution on pressure surface for two grids, model C, $H/D=1.58$, $D=0.25\text{mm}$..	143
Figure A9 – Streamwise row numbers for full vane geometry	144

List of Appendices

A1: Crossflow to Jet Mass Velocity Ratio, G_c/G_j , Extraction and Calculation	135
A2: Discharge Coefficient	136
A3: Katti and Prabhu [24] Streamwise Resolved Correlation Coefficients	137
A4: Segmented Airfoil Wall & the Calculation of the Average Streamwise Resolved Heat Transfer	138
A5: Complete HPT Vane Mesh	140
A6: Crossflow Planes for Full Vane Geometry	142
A7: Mesh sensitivity for HPT Vane models B and C	143
A8: Streamwise Resolved Heat Transfer for Full HPT Vane Geometry	144
A9: Local Nusselt Number Extraction	145

Nomenclature

TIT	Turbine Inlet Temperature	L_c	Characteristic length
D	Jet Diameter	ΔP	Pressure loss %
h	Heat Transfer Coefficient (W/m^2K)	L_e	Entrance Length -nozzle
$R_{1/2}$	Jet thickness – radial distance to $U_{1/2}$	A_j	Jet(s) exit area
$U_{1/2}$	$1/2$ of maximum velocity U_m	A	Total heated area
U_m	Maximum velocity - center line of jet	A_f	Open ratio, A_j/A
U	Velocity – averaged on jet exit		
Z	Vertical length coordinate –from nozzle to target plate		
H	Nozzle to target plate distance		
Nu	Nusselt Number – hd/k or hL_c/K		
Re	Reynolds number – $\rho UD/\mu$		
Pr	Prandtl Number		
r	Radial distance		
C	Curve length		
L, l	Jet orifice length		
X_n	Streamwise Spacing		
Y_n	Spanwise Spacing		
S	Pitch – distance between jet centers		
C_d	Discharge coefficient		
G	Inlet mass flow per unit of heated area		
G_c	Crossflow mass velocity		
G_j	Jet mass velocity		
GCI	Grid Convergence Index		
Y^+	Non – dimensional wall distance		
ρ	Density		
C_p	Specific heat		
k	Thermal Conductivity		
μ	Viscosity		
γ	Ratio of specific heats		

1.0 Introduction

The necessity for efficient thermal control in gas turbines is a direct consequence of the inevitable requirement for higher turbine inlet temperature. The turbine inlet temperature is the most important parameter determining the specific power, specific weight, and efficiency of an aircraft engine (Tamarin [1]). An increase in turbine inlet temperature (TIT) increases the specific thrust of the aircraft engine, but increases the specific fuel consumption. Shown below is the trend of SFC & specific thrust with TIT and compression ratio:

Figure 1.0 – TIT trends [2]

In order to achieve the low optimal compression ratio that will yield lower weight, complexity, and cost, one needs to counterbalance the higher compressor inlet temperature with a higher turbine temperature. Undoubtedly, the choice of these parameters depends strongly on the application and flight regime of the aircraft. High turbine inlet temperatures are very desirable in long range subsonic aircraft and of course, military jets.

As a response to the demand for higher specific thrust and more efficient aircraft engines, the TIT has increased $\sim 1000^\circ\text{K}$ since the 1950's (Han [2]), achieving modern levels of $\sim 2000^\circ\text{K}$. The difficulty has not been in increasing the temperatures but rather making the materials withstand such high temperatures.

Metal temperature capability has increased 468°C in the past 50 years, this progress has had a significant effect on turbine performance. Nonetheless, modern TIT temperatures are still 333°C above the insipient melting temperature of super alloys, and 649°C above a solid uncooled blade (Koff [3]). It is through efficient convection and film cooling that a turbine blade is capable of sustaining such high temperatures.

Cooling of gas turbine components has multiple implications in the overall turbine cycle: turbine work is lost due to extracted air from the compressor; when film cooling is used, aerodynamic mixing losses are incurred; combustor temperature distribution is difficult to shape when significant air is extracted (Jordal [4]). Therefore, an efficient cooling system is one which will yield the maximum blade temperature without having to use large amounts of bleed air.

Service life is heavily dependent on hot component rupture and fatigue. The time on wing (TOW) for the engine, is therefore dependent on the life cycle of the blades, which has been shown through a life sensitivity analysis to depend mostly on the external and internal heat transfer coefficients (Haubert [5]). What determines the heat transfer coefficients is mainly the internal convection and external film cooling. Hence, it is of great importance to optimize the cooling if one wishes to increase blade life, and therefore, reduce yearly engine overhaul costs.

More recently, the gas turbine industry has received significant pressure from environmental organizations. It has become a continuous endeavor to reduce pollutant emissions, specifically NOx. Research into this problem has led to advance combustor concepts which will require more air for combustion, thereby reducing the amount of air for cooling (Lefevbre [6]).

Heat transfer in gas turbines has progressed significantly in the past 60 years, especially in the aero engine application. The aforementioned pressures, whether they may be performance, cost, or environmental, has produced a turbine blade which is considered by many the world's most advance heat exchanger (Koff [3]). It has achieved this status by using a variety of cooling techniques in such a small space: film, ribs, impingement, pin fins, dimples, serpentine roughened passages, transpiration, and swirl chambers. Of concern to this study is the cooling of high pressure turbine vanes which are typically cooled through impingement chambers.

1.1 Impingement Cooling in HPT Vanes

Impingement cooling occurs when a jet of fluid strikes a surface with a higher temperature. The turbulence generated by the jet yields high heat transfer rates (see 2.1). This makes this type of cooling an excellent choice for high heat flux applications such as turbine blades. The high pressure turbine vanes are located at the end of the diffuser downstream of the combustion chamber. At this location, the HPT vanes experience the highest heat flux both through convection and radiation from the combustion gases. Although, recent research has shown that radiation is only dominant for the design of the TBC (Thermal Barrier Coating), and not the active cooling design (Siegel and Spuckler [7]). At the typical 2000°K firing temperatures, the vanes will require approximately ~10% of the core flow to maintain them at a reasonable temperature. The HPT blades only require ~4%, this due to their relative motion which reduces the total temperature (Koff [3]).

An example of impingement cooling inside an HPT vane is shown in Figure 1.1.

Figure 1.1 – Impingement cooling of a 1st stage HPT vane [8]

The blade shown is part of the E³ GE high efficiency engine which served as a technology demonstrator (Halila et al [8]). Clearly shown is the use of two separate impingement chambers, and strategically located film holes around the external surface. The use of two separate impingement inserts and film holes is a necessity for the 1st stage of the turbine, which experiences the highest heat flux. The pressure for each baffle is controlled separately and allows for more efficient use of bleed air where required. Each baffle is manufactured separately and welded into the airfoil via support ribs. Figure 1.2 depicts the impingement inserts for a 1st stage vane.

Figure 1.2 – Impingement inserts for 1st stage Vane, A: Leading edge, B: Aft

The inserts are designed with a geometric pattern of impinging jets. The flow is forced spanwise from the casing of the engine into the root of the blade, and consequently exits the jets to strike the inner surface of the airfoil. For this particular blade, the spent air is used for film cooling of the outer airfoil surface and trailing edge pressure side cooling. The 2nd stage vane does not experience as high heat fluxes as its forward counterpart, and thus, it only requires 1 insert and no film holes to achieve the lower temperature¹. Figure 1.3 depicts the 2nd stage vane for this particular engine.

¹ The desired cooling scheme is heavily dependent on the producer of the engine, but for this particular engine, such a cooling system was found appropriate.

Figure 1.3 – 2nd stage vane with single impingement insert

An attractive characteristic of impingement inserts is their adaptability. Through engine testing, one may find localized regions of high temperature. The insert can be easily modified (drill extra holes) to provide preferential cooling in that region, a capability which is not an easy option for other cooling schemes: film cooling, internal ribs, dimples. The jet diameters encountered in turbine blade applications range in size, but are typically larger than 0.5mm. With the continuous advancement of manufacturing, smaller diameter jets have become an actual possibility, but have yet to be assessed for gas turbine blade applications.

Predicting the heat transfer distribution under an array of jets is of great interest to the turbine blade designer. Such knowledge would allow one to optimize for minimum bleed to reduce impact on turbine cycle, minimum pressure drop which directly affects the pumping power required, or simply, the highest heat transfer rate to attain a higher TIT or reduce mean blade temperature.

1.3 Thesis Objective and Strategy

The chief objective of this study is to *numerically assess through computational fluid dynamics the use of micro-jets ($D < 0.5\text{mm}$) for high pressure turbine vanes*. In doing so, several objectives are accomplished:

- Assess the capability of current commercially available turbulence models in predicting the flow and heat transfer physics of single jet and in-line jet array impingement.

- Model and understand the behavior of an impingement array within a three dimensional realistic model of a turbine vane.
- Compare a current configuration of an HPT vane with one using micro-jets, thus quantifying the differences in heat transfer and flow dynamics.

The thesis is divided into 7 chapters of which chapter 1 is the introduction. Chapter 2 is a comprehensive review of impingement cooling physics and a literature review. Chapter 3 looks at the modeling of a single jet impinging on flat surface. The objective of this chapter is to assure that the commercial package Fluent, and given turbulence models, is capable of replicating the fundamental flow phenomena of impingement cooling. It also serves as a base of preliminary results in numerically assessing a small diameter impinging jet (0.5mm), which has not been well established in current literature. Chapter 4 is a required validation for a simplified version of an HPT vane. No experimental research has been performed on full vane geometry, as such, the validation section models only part of the vane at parameters which can be compared to experimental correlations. In doing so, confidence can be assured for full geometry models. The base vane with macro-jets of $D=0.51\text{mm}$ is then analyzed and discussed in chapter 5. Comparisons are made to the simpler validation vane to understand the differences when the full curved vane is modeled. Chapter 6, HPT vane – Micro Jets, modifies the base vane with smaller diameter jets of $D=0.25\text{mm}$ and compares the flow and heat transfer disparities. Any enhancements or detractions are analyzed and discussed. Finally, chapter 7 concludes the study and offers suggestions for future research work.

1.4 Contributions

At the time of publication, a number of the results and conclusions developed in this study have been written into technical publications. A single impinging micro-jet analysis was published and presented at the ASME Turbo EXPO in Berlin Germany, June 2008. The paper addressed the heat transfer predictions of a micro-jet at low Reynolds numbers and jet to target ratios representative of gas turbine geometries. More recently, a paper entitled “Flow Field and Heat Transfer Predictions of a Semi-Confined Micro-Jet Impinging at Transitional and Low Turbulent Reynolds Numbers” was submitted to Heat and Mass Transfer (Springer) and is currently under review. Lastly, it is the intention of the author to submit for publication a paper encompassing the newer results of the full vane impinging array.

The research contained herein has contributed to current knowledge in the field of impingement cooling for gas turbine blades in several ways:

- The single axisymmetric jet analysis extended the numerical predictions of heat transfer and flow field to lower Reynolds number and micro-diameter jets.
- First study to numerically model a realistic impinging array inside a three dimensional high pressure turbine vane. In doing so, turbulence models were assessed for accurate prediction of heat transfer and crossflow effects.
- Evaluation of micro-jets for application in gas turbine vanes.

2.0 Literature Review and Flow Physics

A comprehensive review of the flow physics and previous research on the subject of impingement cooling is presented in this chapter. The first section, 2.1, reviews the fundamental behavior of single jet impingement. The subsequent section, 2.2, reviews in-line impinging arrays characteristic of gas turbine blade geometry. Finally, section 2.3 provides a literature review of the current numerical methods used to simulate impinging flows.

2.1 Single Jet Impingement

The high heat fluxes dissipated by an impinging jet have made this cooling technique very popular in electronic packaging, industrial drying, micro systems, and of course, gas turbines. Consequently, a great deal of research has been exercised to quantify and understand the behavior of a single jet impinging onto a flat surface. The current analysis studies jets with a circular cross section, which for a single jet renders it axisymmetric. Other types of jets have been studied in literature, such as 2-dimensional jets (slot jets), elliptical jets, and many other arbitrary cross-sections [9, 10, 11, 12, and 13]. For turbine vane applications, the general trend has been to use circular jets for their ease of manufacturing and adaptability (Han [2]); hence, only circular jets are modeled in this study.

Impinging jets can be categorized into two different sections: submerged and un-submerged. A submerged jet is one which discharges into a region filled with the same fluid as that of the jet. An un-submerged jet would be one where the fluid regions differ, such as using water to cool a heated surface in an air filled environment. Gas turbine vanes are cooled with bleed air from the compressor, where the jets discharge into an air filled environment, therefore, of concern to this study is a submerged impinging air jet. Furthermore, an impinging jet can be confined, semi-confined, or unconfined. An unconfined jet is one where the fluid is allowed to discharge in all directions, radially and vertically. The presence of a top wall at the nozzle level renders it semi-confined, where the spent air can only exit in the radial direction. The presence of a side and top wall forces the spent air to exit in only one direction, hence, a confined setup. Figure 1.1 clearly illustrates the situation for a typical gas turbine vane: the baffle inside the airfoil serves as a pressure chamber and as well as a top confining wall when the fluid discharges and strikes the inner airfoil surface. Accordingly, all impingement occurring inside turbine vanes can be categorized as submerged and semi-confined.

Figure 2.0 illustrates the different flow regimes which occur when a submerged semi-confined air jet strikes a flat surface.

Figure 2.0 – Single impingement flow regions and definitions

When the fluid is discharged from the nozzle, a free jet is formed. The free jet is characterized by its thickness and spreading rate, which is dictated by the exchange of momentum with the ambient. This momentum exchange induces mixing and entrainment of ambient air into the jet core itself, which as it can be seen from Figure 2.0, expands the jet boundary. The velocities near the boundary of the jet begin to degrade due to loss of momentum almost immediately after the jet is discharged. The peak velocity U_m along the center line of the jet remains constant for a length until the mixing has penetrated up the center line. The length for which the peak velocity remains constant has been named the “potential core” of the jet, and is typically found near $\sim 5D$ (Garimella and Rice [14], Martin [15]). Past the potential core height, the peak velocity begins to degrade at rate proportional² to $1/Z$ (Gardon and Akfirat [16]). Near the target surface, the jet will begin to decelerate and deflect radially continuing as a radial wall jet. For axisymmetric jets, the boundary layer generated in the stagnation region will exhibit a minimum thickness near $r/D=0.5$, and from then on it will continue to thicken [16] with increasing radial distance. The wall jet velocity is degraded due to wall friction and mixing on the top. The top confining wall induces spent air recirculation which can produce toroidal vortices in the vicinity of the stagnation region.

² Where “Z” is the vertical distance measured from the top confining wall towards the target plate

Single jet impingement has always been a challenge to predict numerically (see 2.3) because of the many different types of flow regimes encountered, that is: free jet, stagnation, wall jet, and recirculation zones.

From the description of the flow given above it is expected that the maximum heat transfer would occur in the stagnation region directly under the jet. As the boundary layer thickens, the heat transfer would monotonically decrease in the radial direction. This yields a typical “bell” shaped curve for the heat transfer distribution under an impinging jet, Figure 2.1 illustrates it:

Figure 2.1 - Typical heat transfer distribution under an impinging jet

Jambunathan et al [17] compiles a comprehensive review of experimental work for single jet impingement. The work revealed that most correlations for the average heat transfer obey the following form:

$$\overline{Nu} = f(Re, \frac{H}{D}, \frac{R}{D}, Pr) \quad (2.1)$$

Equation 2.1 indicates that the average heat transfer is mainly a function of the Reynolds number (based on the diameter of the jet), the non dimensional jet to target distance H/D , the non dimensional radial distance examined R/D , and the Prandtl number. Martin [15] obtained a correlation which is known to compare well with most experimental work. For a single circular jet:

$$\frac{\overline{Nu}}{Pr^{0.42}} = G\left(\frac{r}{D}, \frac{H}{D}\right) F_1(Re) \quad (2.2)$$

Where:

$$F_1 = 2Re^{1/2}(1 + 0.005Re^{0.55})^{1/2} \quad (2.3)$$

$$G = \frac{D}{r} \left(\frac{1-1.1D/r}{1+0.1(\frac{H}{D}-6)\frac{D}{r}} \right) \quad (2.4)$$

Suggested use is constricted to $2000 \leq Re \leq 400000$, $2 \leq H/D \leq 12$, and $2.5 \leq r/D \leq 7.5$. Huang and El-Genk's [18] experimental work extends the applicable range for a single circular jet to: $1 \leq H/D \leq 10$, $0 \leq R/D \leq 10$, and $6000 \leq Re \leq 60000$. Their correlation is given by:

$$\overline{Nu} = Re^{0.76} Pr [a + b \left(\frac{H}{D}\right) + b \left(\frac{H}{D}\right)^2] \quad (2.5)$$

Where:

$$a = 10^{-4} [506 + 13.3 \left(\frac{R}{D}\right) - 19.6 \left(\frac{R}{D}\right)^2 + 2.41 \left(\frac{R}{D}\right)^3 - 9.04 \times 10^{-2} \left(\frac{R}{D}\right)^4] \quad (2.6)$$

$$b = 10^{-4} [32 + 24.3 \left(\frac{R}{D}\right) - 6.53 \left(\frac{R}{D}\right)^2 - 0.694 \left(\frac{R}{D}\right)^3 + 2.57 \times 10^{-2} \left(\frac{R}{D}\right)^4] \quad (2.7)$$

$$c = -3.85 \times 10^{-4} \left[1.147 + \left(\frac{R}{D}\right) \right]^{-0.0904} \quad (2.8)$$

A recent study by Hoffman et al [19] uses thermography to measure the heat transfer on the impingement plate. Their study, as well as others, reveals insensitivity to H/D when it is smaller than 6 diameters. As such, their correlation does not include jet to target spacing:

$$\overline{Nu} = Pr^{0.42} [Re^3 + 10Re^2]^{0.25} 0.055 \cdot \frac{1 - \exp(-0.025(\frac{R}{D})^2)}{0.025(\frac{R}{D})^2} \quad (2.9)$$

Equation 2.9 is valid for: $14000 \leq Re \leq 230000$, $0.5 \leq H/D \leq 10$, $0 \leq R/D \leq 8$.

From the shown equations it can be seen that the average Nusselt number has been well studied and correlated. Yet, the local heat transfer distribution has proven difficult to correlate due to its sensitivity to various parameters such as: velocity profile – nozzle configuration, thermal condition on target plate, confinement setup, and jet fluid-ambient temperature difference. For this reason, local heat transfer correlations are scarce and unpractical (too many variables). Nevertheless, Hoffman et al [19] provide a quick and practical correlation for general purposes:

$$Nu = 0.055 [Re^3 + 10Re^2]^{0.25} \cdot Pr^{0.42} \cdot e^{-0.025(\frac{r}{D})^2} \quad (2.10)$$

Where the same range of validity applies as equation 2.9.

The maximum heat transfer does not necessarily occur under the jet, as such, the behavior of stagnation point heat transfer is also well studied. Garimella and Rice [14] provide the following correlation for stagnation point heat transfer for a circular jet:

$$Nu_{sg} = 0.492 Re^{0.585} Pr^{0.4} \left(\frac{H}{D}\right)^{0.024} \left(\frac{l}{D}\right)^{-0.09} \quad (2.11)$$

The range of validity: $1.59\text{mm} \leq D \leq 6.35\text{mm}$, $4000 \leq Re \leq 23000$, $1 \leq H/D \leq 5$, $0.25 \leq l/D \leq 12$.

Increasing the velocity of the jet, or increasing the Reynolds number, accordingly increases the heat transfer on the plate. For jet to target distances greater than $H/D = 5$, the bell shaped curve illustrated in Figure 2.1 simply shifts upwards. With increasing jet to target distance and increasing Reynolds numbers, the Nusselt curve sharpens. Low H/D ratios and Reynolds numbers typically yield more smooth and rounded curves. This is evident in the work of Goldstein and Behbahani [20].

Decreasing the jet to target distance, H/D , has a similar effect to increasing the Reynolds number - increasing the overall heat transfer. Yet, the relationship is more complex due to inner peaks and secondary maxims which are formed at low H/D values. It has been well established for single circular jet impingement that at $H/D < 5$, secondary maxims occur in the region near $r/D \approx 2$. Such a peak has been observed experimentally [15, 16, and 19]. Figure 2.2 illustrates this phenomenon:

Figure 2.2 – Nusselt distribution for a single circular jet with $H/D < 5$

The secondary peak illustrated in Figure 2.2 has been attributed to the transition of the boundary layer from a laminar to turbulent regime. The work of Gardon and Akfirat [16] was one of the first experimental studies assessing the secondary peaks under impinging jets. In the stagnation region, favorable pressure gradients exist which help stabilize the laminar boundary layer. Upon exiting the jet deflection region, the pressure gradients disappear which promote the transition to a turbulent

boundary layer. With increasing jet to target distance, the jet turbulence increases (due to mixing and entrainment of the ambient air), as such, the transition is diminished due to a ready present high levels of turbulence in the jet/stagnation region. Therefore, the secondary peaks disappear at higher H/D ratios, usually $H/D > 5$ [16].

Inner peaks at $r/D = 0.5$ have also been reported in literature for axisymmetric jets (16, 21, 22). This leads to the maximum heat transfer to occur at the inner peak rather than at the stagnation point. As opposed to the secondary maxima explain earlier, the inner peak is not attributed to boundary layer transition, but rather to boundary layer thickness. Axisymmetric jets have a minimum thickness near $r/D = 0.5$ which leads to a higher heat transfer at that location [16, 21, 22]. The inner peak becomes more pronounced with higher Reynolds numbers, and diminishes with increasing H/D .

The effect of confinement has been studied experimentally by several authors (Obot et al [23], Behnia et al [24], Fitzgerald and Garimella [25], Li et al [26], Koseoglu and Baskaya [27]). The results indicate that the top confinement wall reduces the entrainment of ambient air into the jet, decreasing the overall average heat transfer by 5-10% (Obot et al [23]). Behnia et al [24] studied the effect of confinement for several H/D ratios. The analysis revealed that confinement only affects the local heat transfer distribution at very low H/D ratios, $H/D < 0.25$, with the main decrease occurring downstream of the stagnation region. Fitzgerald and Garimella [25] studied the behavior of the toroidal vortices generated for a top confined impingement setup. The findings indicate that the center of the vortices move radially outward with increasing Reynolds number and jet to target distance. These results were confirmed with the work of Li et al [26]. The recent work of Koseoglu and Baskaya [27] compared the flow field for circular and elliptical semi-confined impinging jets. Toroidal vortices were observed at $H/D = 2$ for both types of nozzles, with flow field differences disappearing with increasing jet to target distance.

2.2 Inline Impinging Arrays

The flow behavior occurring inside an HPT vane is fundamentally illustrated in the diagram below:

Figure 2.3 – Impingement flow inside an HPT vane [31]

The flow enters from the casing of the engine (into the page) and is forced out from the orifices of the baffle to strike the inner surface of the vane. For impingement only vanes, the flow is forced to exit at the trailing edge, which, as it can be seen from Figure 2.3, deflects the downstream jets. Experimental work in gas turbine blades models the vane as a flat plate, which Figure 2.3 illustrates is possible in regions of low curvature. The work of Van Treuren [28, 29], Kercher and Tabakoff [30], Florschuetz et al [31,32], and Bailey and Bunker [33] are excellent representatives of the typical experimental research studying the behavior of impinging arrays for gas turbine blade applications. The experimental setup is usually of the following nature:

Figure 2.4 – In-line impinging array representative of a gas turbine vane [28]

Confined on one side, the flow exiting from the jets is forced to exit in one direction, thereby duplicating the flow encountered in a vane. The main differences between the actual vane and that of the experimental setup are the flat geometry, and the direction of flow entry. The mass flow into the array is aligned with the jet discharge direction; this is not the case with an actual vane, where the flow enters the plenum perpendicular to the jet discharge direction.

An impinging array can be of two types: in-line and staggered. An in-line array is shown in Figure 2.4, and the name dictates the relative position between the jets. Staggered arrays can have many different patterns of jet positions. It has been shown (Han [2]), that for the range Reynolds numbers and geometric parameters pertaining to gas turbine blades, in-line arrays over perform staggered ones. This is believed to be the case due to the crossflow being channeled through the jets themselves, thereby accelerating and augmenting forced convection through the impingement channel. Crossflow in staggered arrays is disturbed by the misaligned jets, thereby yielding lower heat transfer. Nevertheless, staggered arrays are very often used in many other applications, including certain locations in gas turbines. The current study focuses only on in-line impinging circular jet arrays, which are the most common in gas turbine blade applications.

In-line arrays are described by a set of geometric parameters - as shown in Figure 2.4. The streamwise spacing, X_n : spacing between jet centers in the direction of the exiting flow. Spanwise spacing, Y_n : spacing between jet centers in the spanwise direction (hub to tip in the actual vane); the jet to target spacing H/D , or Z/D : spacing between the jets and the impingement surface; L/D , the ratio between the impingement plate thickness and the diameter of the jet.

The streamwise jet velocity distribution has been observed to increase from the initial row of jets to the exit row. This has been attributed to the increase in crossflow as more air is introduced into the impingement channel via the downstream jets. With the increasing crossflow velocities, the downstream jets effectively increase in exit velocity and conversely, the pressure decreases across the array. Thus, the heat transfer under the jets depends heavily on a careful balance between the higher exit speed of the downstream jets, and the higher crossflow which renders them more susceptible to deflection. The experimental work of Florschuetz et al [31] assessed the different parameters which affect the crossflow distribution across the array. A crossflow to jet mass velocity ratio was derived from a flow distribution model which agreed with their experimental data. It is given by:

$$\frac{G_c}{G_j} = \frac{1}{\sqrt{2}C_D} \frac{\sinh \beta(x/x_n - 0.5)}{\cosh \beta(x/x_n)} \quad (2.12)$$

Where

$$\beta = \frac{C_D \sqrt{2} \left(\frac{\pi}{4}\right)}{\left[\frac{Y_n}{D}\right] \left[\frac{Z}{D}\right]} \quad (2.13)$$

G_c is the crossflow mass velocity, or the mass flow divided by the cross-sectional area of the channel. G_j has the same definition but it is calculated for jet mass flux, the reader is referred to the appendix A1 for a detailed illustration and explanation. Equations 2.12 and 2.13 depend on the average discharge coefficient of the plate, C_D , which for this numerical study is calculated using isentropic relations (appendix A2).

The Florschuetz et al [31] model leads to the result that the G_c/G_j crossflow ratio is independent of the streamwise spacing of the array, but is heavily dependent on the jet to target and spanwise spacing. Their analysis also revealed that the larger the $(Y_n/D)*(Z/D)$ factor, the more uniform the jet and crossflow distribution will be across the array. A highly non uniform distribution is obtained (increase in jet velocity across the array) when $(Y_n/D)*(Z/D) = 4$, compared to an essentially flat distribution for $(Y_n/D)*(Z/D) = 24$. What this entails is that the crossflow experienced by the jets decreases with an increase in jet to target spacing (larger channel height), and larger spanwise spacing.

The heat transfer under an impinging array is typically resolved in the streamwise direction. That is, averaged across the span of the array and plotted for each streamwise row. Key parameters which influence the heat transfer rate are: the Reynolds number, the crossflow ratio, the streamwise and spanwise spacing, the jet to target spacing, and the exit/entrance directions. The Reynolds number has the same effect on the array as that of a single jet- increasing the average and local values under the jets with increasing exit velocities. The general relationship between the overall average Nusselt number and Reynolds is that of a power expression [28-33] such as:

$$\overline{Nu} = Re^m \quad (2.14)$$

Where the exponent “m” is dependent on the geometrical properties of the array. In this study, several correlations will be used to compare the numerical results to that of experimental work. From the work of Florschuetz et al [31], the following correlation for the streamwise resolved Nu is obtained:

$$Nu = A Re_j^m \left\{ 1 - B \left[\left(\frac{Z}{D} \right) \left(\frac{G_c}{G_j} \right)^n \right] \right\} Pr^{\frac{1}{3}} \quad (2.15)$$

$$A, m, B, n = C \left(\frac{x_n}{D} \right)^{n_x} \left(\frac{y_n}{D} \right)^{n_y} \left(\frac{z_n}{D} \right)^{n_z} \quad (2.16)$$

For equation 2.16, Table 2.1 summarizes the required coefficients:

Table 2.1 – Coefficients for equation 2.16, Florschuetz Nu correlation

	C	n_x	n_y	n_z
A	1.18	-0.944	-0.642	0.169
m	0.612	0.059	0.032	-0.022
B	0.437	-0.095	-0.219	0.275
n	0.092	-0.005	0.599	1.04

The experimental work of Bunker and Bailey [33] extends the previous correlation to denser square arrays (same streamwise and spanwise spacing) and arrays of larger extent. Their analysis revealed that the denser arrays inhibit significant channel flow behavior due to high crossflow values at small Z/D and jet to jet spacing values. A correlation for the streamwise resolved heat transfer is obtained:

$$\begin{aligned} \overline{Nu} = & 47.1 - 5.5 \frac{X}{D} + \frac{Z}{D} \left(7.3 - 2.3 \frac{Z}{D} \right) + Re \left(4 \times 10^{-3} - 1.3 \times 10^{-4} \frac{X}{D} - 1.5 \times 10^{-8} Re \right) \\ & + \frac{G_c}{G_j} \left(61.2 - 13.7 \frac{X}{D} - 28 \frac{Z}{D} \right) \end{aligned} \quad (2.17)$$

The experimental work of Katti and Prabhu [34] investigates the effect of spanwise spacing on the heat transfer distribution. Their analysis reveals that the non-uniformity in jet flow and non-linearity of the channel flow increases with decreasing jet to target spacing and spanwise pitch. An optimal spanwise spacing value of $4D$ was found to yield the highest average heat transfer rate for a given pressure loss. A correlation for the average streamwise resolved Nusselt number was obtained:

$$Nu = a_o (Re^{a_1} Pr^{0.33}) \left(\frac{G_j}{G_j} \right)^{a_2} \left(1 - b_o \frac{G_c}{G_j} \right)^{b_1} \quad (2.18)$$

A list of the required coefficients in equation 2.18 can be found in the appendix A3.

Brevet et al [35] consider a row of impinging jets for optimization. The analysis revealed an optimal jet to target distance for maximum heat transfer between $2D$ and $5d$, with $3D$ being considered a good initial design value. Likewise to Katti and Prabhu [34], an optimal spanwise spacing of $4D$ was also found. Van Treuren [28,29] studied three different jet to target ratios, $1D$, $2D$, and $4D$. The results indicate an optimal point for maximum heat transfer between $2D$ and $4D$. The general consensus found in the aforementioned experimental work is that of a continuous decreasing average heat transfer at any jet to target spacing larger than $5D$. As such, in gas turbine blade applications, the designer sets the lowest possible jet to target spacing, given manufacturing considerations (Zuckerman and Lior [36]).

Huang et al [38] investigated the effect of spent air exit direction. The results indicated that the streamwise resolved heat transfer is significantly affected by the relative direction of the flow entry and exit. Maximum heat transfer is obtained when the spent air is allowed to exit in two directions. This was attributed to a decrease in crossflow ratio since all the accumulated spent air does not deflect all downstream jets. Three different entry/exit orientations were tested, where the flow path is still considered two dimensional. In other words, the entry, jet discharge, and exit all occur in two planes. In a real vane, all directions are perpendicular to each other, that is, the flow enters in the spanwise direction, the jets discharge perpendicularly, and the spent air exits streamwise. No experimental or numerical research has been performed which uses a fully three dimensional entry/exit orientation.

Gao et al [39, 39] studied the effect of linearly stretching the spanwise and streamwise spacing. In doing so, replicating more realistic turbine geometry where the impingement holes are directed towards hot spots. The heat transfer distribution indicated that the jet to jet interaction decreased with downstream length, and that the in-between jet regions increased in heat transfer. The crossflow effect was clearly evident with the widening of heat transfer zone behind the jets, as oppose to symmetrical around the stagnation region.

The choice of jet diameter is usually dependent on the design pressure loss, and the number of holes per unit of heated area (Andrews et al [40]). In experimental work, the effect of jet diameter has not been investigated directly. Andrews et al [40], investigate the effect the number of holes has on the average heat transfer, that in itself dictated the jet diameter, which varied from 0.64mm to 4.70mm . Their analysis did not assess the local Nusselt distribution, only the overall heat transfer. Florschuetz et al [31] experimental work varied the jet diameter in order to accommodate a certain number of jets per area. Their smallest jet had a diameter of 0.635mm . The effect of jet diameter was

only compared for the pressure transverses across the array, and not for the heat transfer rate on the plate. Van Treuren et al [28, 29] did not vary the diameter of the jets; instead, a large diameter of 5mm was used. Bailey and Bunker [33] vary the diameter of the jets to accommodate different jet spacing values – the diameter of the jet was set to 5.08mm or 10.16mm, which are relatively large values. Likewise, the experimental work of Brevet et al [35], Huang et al [37], and Katti and Prabhu [34], all evaluate the local and average heat transfer values using large diameter orifices, $D > 3.1\text{mm}$. In the experimental work mentioned, the effect of jet diameter on the hydrodynamics and heat transfer has not been studied within constant array geometry. Yet, single jet results, Garimella and Nenaydykh [41], observe higher heat transfer at a constant Reynolds number for a smaller diameter nozzle than a larger nozzle. As such, the diameter of the jet has not been properly investigated for impinging arrays, particularly in gas turbine blade applications.

Several experimental studies have revealed great potential for micro-jets. Fabri et al [42] experimentally compare a liquid micro-jet array and spray to cool high power density electronics. They found the micro-jet array superior to the spray since it required less power for unit of power removed. Phalnikar et al. [43] studied supersonic impinging micro-nozzles. Their experimental work revealed lower spreading rates for micro-jets, and thus higher concentration of cooling fluid in the stagnation region. Leland and Klasing [44] constructed a micro array device to cool high power micro-electronic packages. Although successful in cooling, the results indicated that a more accurate formulation must be used for micro sized arrays. Overall, Micro-jets $< 0.5\text{mm}$, have shown tremendous potential in micro-electro-mechanical systems (MEMS), but this is yet to be fully explored for gas turbine blades, where higher pressure values can be readily accommodated to drive the flow through the smaller jets.

2.3 Numerical Analysis for Impinging Flows

Impinging flows have proven quite difficult to numerically model due to their complex flow regimes which have been mentioned before: free jet, stagnation region, wall jet, and recirculation/mixing for a confined setup.

The numerical work of Behnia et al [24] evaluates the flow and heat transfer predictions for unconfined and confined axisymmetric jet impingement. Two eddy viscosity turbulence models were used to assess the predictions: the standard k-epsilon and the newer v^2 -f model. The results indicated that the k-epsilon model was incapable of replicating the flow features, with the main deficiency being

the over-prediction of turbulent kinetic energy in the stagnation region, thereby yielding significantly higher heat transfer. The use of wall-functions or damping functions did not help alleviate the high stagnation values obtained. The v^2 -f model was capable of replicating with good agreement the heat transfer and flow features of the axisymmetric jet at various H/D and Re values. The same conclusion regarding the k-epsilon model was obtained by the work of Cooper et al. That is, an over-prediction of stagnation values which are due to the high levels of turbulent kinetic energy.

Coussirat et al [45] assess a variety of eddy viscosity models by comparing experimental data to the prediction of field velocities and heat transfer for an unconfined single round impinging jet and a planar jet. Their analysis include: the two equations K-epsilon family (standard, realizable, and RNG), one equation Spallart –Almaras, and the four equation v^2 -f model. A two layer k-epsilon model with a different a length scale in the near wall region is used, this alleviates some of the over-prediction of turbulent kinetic energy which was obtained in the earlier mentioned numerical work. The results indicated that average values for heat transfer and velocities were all well captured. Strong sensitivity was obtained for all turbulence models in capturing the stagnation heat transfer, which varied heavily depending on the H/D ratio and turbulence nozzle exit profiles. Both the K-epsilon family and the Spallart-Almaras models obtained similar results, making the Spallart-Almaras a good choice for initial results as it has lower computational costs. The v^2 -f model obtained the best overall agreement with the heat transfer results, but the distinction was only evident at lower Reynolds numbers. Only the RNG K-epsilon and the v^2 -f models were capable of replicating the secondary maximum due to the transitional boundary layer, although, the location was much better captured with the v^2 -f model.

Spring et al [46] assessed the numerical predictions of a single circular jet impinging through crossflow. The turbulence modeling assessment revealed that the K- ω SST model, a blend of the k-epsilon and k- ω model, obtained the best agreement with experimental data. Zhang et al [47] compare several turbulence models on the prediction of confined single nozzle impingement cooling for application in micro-electronic packaging. Turbulence kinetic profiles were extracted to show the benefit of using the RNG K-epsilon model, which includes a damping function to alleviate the improper modeling of turbulent dissipation in regions of high strain rate (stagnation region). The results indicate that the RNG k-epsilon model is preferable to other variances of the K-epsilon model, and even more so than the advanced 7 equation Reynolds Stress Model. The same conclusion was obtained with the work of Shen et al [48] who analyze the effect of impingement angle for a confined jet. In the latter study, the

K-epsilon RNG was found to yield a lower average deviation than the K ω -SST, although, the K ω -SST obtained better agreement for the non-orthogonal jets.

Kumar and Prasad [49] analyze the predictions of a row of jets impinging on concave surface, thus replicating leading edge impingement cooling. Their results indicate K ω -SST being a better predictive model for impingement flow on concave surfaces. In comparison to experimental correlations, maximum deviation for the average Nusselt number was 18%, attaining good agreement for the complex flow modeled. El-Gabry and Kaminski [50] model a single streamwise column of 7 jets forced to exit in 1 direction, thus replicating part of the experimental setups mentioned in 2.2. Their analysis compared the Standard k-epsilon model (with the two layer near wall treatment), and the low Reynolds number Yang-Shi model. Better agreement was obtained in the in-between regions of the jets with the Yang-Shi model, but it predicted higher stagnation values than the K-epsilon and the experimental data. No decisive conclusion was made on the better performing model, but both models were capable of replicating the crossflow effect on the impinging jets.

A thorough review of numerical studies assessing different turbulence models and their capability in predicting heat transfer for impinging flows was done by Zuckerman and Lior [36]. Not counting DNS, or LES³, The v^2 -f model ranked 1st, K ω -SST ranked 2nd, and the K-epsilon family (standard, RNG, realizable) 3rd, tied with algebraic stress models.

On the type of mesh to be used, both hybrid (tetrahedral and hexahedral) and purely hexahedral grids have been used successfully in several impinging flow studies. Single jet studies typically use structured hexahedral grids [24,46,51], while more complicated three dimensional models have relied on tetrahedral elements with a boundary layer mesh in the near wall region [49,52].

2.4 Summary

To the best of the author's knowledge, micro-jet impingement cooling has not been experimentally or numerically studied for application in gas turbine blades, both in single jet setup and full impinging arrays. In most studies, the jet diameter is varied to acquire different array parameters, and thus, the full potential of decreasing the jet diameter below 0.5mm (500 μ m) has not been properly explored for the gas turbine blade application. Micro-jets have shown tremendous potential for

³ Direct Numerical Simulation (DNS) and Large Eddy Simulation (LES). Considering the scope and timeline of this thesis, advance numerical simulations requiring very fine grids were not considered.

electronic cooling, and thus, it is expected that heat transfer enhancements should as well be obtained for turbine blades, noting that higher pressure flow is readily available in the latter application. Impingement studies simplify the vane geometry via a flat plate, thereby assuming that vane curvature should play no significant role. No experimental or numerical study has assessed the heat transfer distribution for full blade geometry, but a wealth of information is available for flat or leading edge-only regions. In conclusion, a gap exist in current literature in which micro diameter impinging jets have yet to be studied on their capability to enhance heat transfer on a full three dimensional gas turbine blade.

3.0 Numerical Modeling of a Single Axisymmetric Impinging Jet

In order to gain a better insight into the commercial CFD code Fluent, a basic model of a single jet impinging onto a flat plate is solved. This provides a basic understanding of the flow behavior of a semi-confined impinging jet. Following the validation and assessment of different turbulence models for this setup, a more realistic geometry with several jets can be analyzed.

3.1 Geometry and Modeling

A single circular jet impinging onto a flat surface is a simple geometric problem which is depicted below:

Figure 3.1 – Axisymmetric impinging jet geometry

The chosen control volume for this model is a $10D \times H$ cylinder, where the entire bottom surface is the target plate to be cooled. The initial height was set at $6 \times 500\mu\text{m}$ or 3mm which yields an H/D ratio of 6, and the nozzle diameter was fixed at $500\mu\text{m}$ (0.5mm). The circular geometry was chosen to keep the symmetry of the impinging jet. Preliminary results with a square plate showed uneven temperature distributions which can be misleading. To obtain the effect of changing the H/D ratio, the height of the cylinder was altered to obtain the desired jet to target spacing. Altering the height and not the diameter of the nozzle, kept the exit velocity constant across all H/D cases iterated.

The impingement model is semi-confined, where the top surface is solid and all sides are pressure outlets. The semi-confined setup was chosen since all HPT vane impingement cooling is

constrained by the plenum itself. For validation purposes, the flow is channeled through a long cylindrical nozzle to obtain a fully developed profile at the exit. The entrance length for fully developed flow can be calculated from the following (White [53], turbulent: $Re > 2300$):

$$\frac{Le_{Lam}}{D} = 0.06Re \quad \frac{Le_{Turb}}{D} = 4.4Re^{1/6} \quad (3.1)$$

The range of parameters for the single jet analysis spans from a low jet to target spacing of 1D to a relatively high value of 6D, and Reynolds numbers 2000 to 6000.

3.1.1 Mesh Creation

Due to the axisymmetric nature of the single jet model, only a quarter of the full model (Figure 3.1) is meshed. Two symmetry conditions are used on each axis, thus producing the full geometry. The mesh is fully hexahedral and structured. Gambit v2.1.6 is used with a cooper scheme: the top face mesh is extruded to fill the volume mesh. Edges are set with a prescribed pattern of nodes so as to increase in density near the target impingement wall, thus assimilating a boundary layer mesh and assuring a low y^+ value. Figure 3.2 below illustrates the mesh used for a jet to target spacing of 6D (nozzle removed for clarity):

Figure 3.2 – Singlet jet mesh, $H/D=6$

3.1.2 Numerical Setup and Boundary Conditions

All cases were run with constant density air (incompressible & constant properties). For the typical Reynolds numbers tested <6000 there was no noticeable effect in iterating with compressible and varying viscosity air. All walls in the model except the target plate were kept adiabatic. Convergence criteria for the residuals were set at E-04, except for the energy equation which was set at E-06. Momentum and energy solutions were iterated with a 2nd order upwind discretization method, and turbulence model variables were left as 1st order. The target plate was given a constant temperature boundary condition of 600°K. The ambient and jet air temperatures were kept at the default value of 300°K. Keeping the ambient and jet temperatures equal and relatively low, allows the use of the jet temperature to calculate the heat transfer coefficient. Goldstein and Behbahani [20] showed that if the ambient and jet temperatures differ greatly, the adiabatic plate temperature must be used to calculate the heat transfer coefficient. Therefore, for this analysis the heat transfer coefficient is obtained from

equation 3.2. Finally, all cases were iterated using Fluent 6.3.26 with a segregated solver, implicit formulation and steady flow. Unless otherwise specified, all the above conditions should be assumed for the case being analyzed.

The usual definition of the heat transfer coefficient is used:

$$h = \frac{q''}{T_s - T_j} \quad (3.2)$$

Where the “ T_s ” is the temperature of the target surface (600°K), and T_j , the mean jet temperature (300°K). The non-dimensional form of the heat transfer coefficient, the Nusselt number, is calculated as follows:

$$Nu = \frac{hD}{k} \quad (3.3)$$

“ k ” is the thermal conductivity of air at 300K. A list of the fluid properties used in the single jet study is summarized in Table 3.1:

Table 3.1 – List of air properties at 300°K for the single jet study

Density (kg/m ³) - ρ	1.225
Specific Heat (J/kg°K) - C_p	1006.43
Thermal Conductivity (W/m°K) - k	0.0242
Viscosity (Kg/ms) - μ	1.79E-05

3.2 Mesh Sensitivity

A preliminary simulation was executed in order to obtain the grid sensitivity, and to determine its discretization error. As the error is expected to increase with Reynolds number, the highest Re is selected ($Re_D=6000$, $H/D = 6$) to evaluate the average spanwise Nusselt number, defined as:

$$\overline{Nu} = \frac{1}{x} \int_0^x Nu \, dx \quad (3.4)$$

Starting with an arbitrary grid, two other grids are successively created by using a refinement factor of $r = 1.3$. The average Nusselt numbers obtained for the three grids are given in Table 3.2:

Table 3.2 – Average Nusselt numbers for three grids

	Coarse	Intermediate	Fine
$\overline{Nu}$	24.83	26.61	27.47

As it can be seen the Nusselt values are asymptotic as the grid gets finer. This allows for the use of Richardson's extrapolation theory and the Grid Convergence Index (GCI) suggested by Roache [54, 55]. Table 3.3 summarizes the calculated values for the refinement study:

Table 3.3 – Mesh sensitivity analysis summary

Order of Convergence	2.79
Zero Spacing Nu Value	28.27
GCI Coarse %	0.078
GCI Fine %	0.036

The grid sensitivity analysis reveals that the average Nusselt number can be predicted with the fine grid at a maximum discretization error of 0.036%. Further refinement of the fine grid produces an insignificant improvement which only increases computational time. The zero spacing value obtained by using Richardson's extrapolation confirms that small improvement which is achieved if further refinement takes place. It is expected that the local Nusselt numbers will have higher discretization errors, and furthermore, modeling errors cannot be accounted for in this refinement study. For that reason, the local distribution at $H/d=6$, $Re=6000$ is compared for the three grids:

Figure 3.3 – Local Nusselt distribution for three grids at $H/D=6$, $Re=6000$

As it can be seen, all grids predict the same distribution, with very little deviation occurring between the fine and medium grid.

In order to use enhanced wall treatment for all turbulence models, the y^+ value was kept below 5, as recommended from [59]. The Maximum y^+ value for the fine grid is 3.65, thus making it viable to use enhanced wall treatment.

3.3 Turbulence Model Selection

To evaluate the different turbulence modes, the local distribution is compared to the experimental work of Angioletti et al [51]. For the average heat transfer values, Martin [15] is used since most recent studies compare well with its correlation (Eq.2.5). Figure 3.4 plots the predicted distributions for all turbulence models and the experimental data of [51]:

Figure 3.4 – Local Nusselt Distribution, turbulence models and experimental data from [51], $H/D=4.5$, $Re=4000$

The experimental values end at $r/d = 3$, but it is just enough to see the agreement with the numerical results. It is clear from Figure 3.4 that it is only the stagnation region which is not accurately predicted. None of the turbulence models are capable of replicating the flat maxima occurring in the

stagnation region, with only the RNG model coming close to the actual values. The k- ω SST agrees well with the data downstream of $r/d=1$, but severely under predicts the stagnation region with the typical saddle shape that you see with lower H/D values. The k- ω model under predicts the overall local distribution, but does have a similar geometric agreement with the stagnation region.

The average Nusselt number and deviations with the experimental correlations are summarized in Table 3.4:

Table 3.4 – Average Nusselt number and deviations with experimental correlations $H/D=4.5$, $Re=4000$

Turbulence model	$\overline{Nu}$	Martin $\overline{Nu}$	% deviation	Angioletti $\overline{Nu}$	% deviation
K-ϵ	14.845	20.812	28.67	17.0411	12.89
K-ϵ RNG	15.906		23.57		6.66
K-ω	11.991		42.38		29.63
K-ω SST	14.581		29.94		14.44

The difference between the numerical results and the Martin [15] correlation can be attributed to the small radial distance ($r/D=3$) which was used for comparison with Angioletti [51]⁴. Nevertheless, it is clear that the RNG turbulence model captures the average Nusselt value, and the local Nu distribution the closest. Other cases which were iterated, presented similar results to Figure 3.4. This indicates that the RNG k- ϵ model best suits this impingement setup for the set of parameters solved in this study. This choice is not unprecedented, the numerical work of Shen et al [48], and Zhang et al [47] both found better agreement for impinging flows with k- ϵ RNG than other commercially available turbulence models.

3.3 Results

3.3.1 Impingement Flow Field

A key characteristic of semi-confined impingement is the recirculation pattern formed in the vicinity of the jet (Figure 2.0). Understanding the behavior of the recirculation pattern with increasing Reynolds number and jet to target spacing is of importance, as this affects the heat transfer, and jet to jet interaction in an impinging array. The capability of the turbulence model capturing the trends with changing parameters in the recirculation pattern must also be verified. Figure 3.5 illustrates the effect of increasing Reynolds number on the toroidal structure and its center:

⁴ The deviation reduces with increasing radial distance – Evident in the average heat transfer results, section 3.4

Figure 3.5 – Streamlines illustrating the toroidal vortex formed at $H/D=1$, and Reynolds numbers 2000-6000

At Reynolds number 2000 and 3000, the toroidal structure is not fully defined. A large oval is formed rather than a circular center which is seen at higher Reynolds numbers. As the Reynolds number is increased to 3000, the oval is pinched in the middle, and the toroid moves towards the jet. At $Re = 4000$, the transition of the center is clearly illustrated as the right center diminishes. Further increase of the Reynolds number moves the center of the toroid radially outward. The same trend for toroidal movement at a constant jet to target spacing was found by Fitzgerald and Garimella [25], and Li et al [26]. For $H/D = 1$ and $Re=3000$ to 6000, the outward movement of the toroid center is nearly linear, moving towards the outlet at a $0.267(r/D)$ per 1000 Re increase. At these low Reynolds numbers, no significant vertical movement is seen by the toroid.

Figure 3.6 illustrates the effect on the recirculation pattern when the jet to target spacing is changed:

Figure 3.6 – Streamlines illustrating the toroidal vortex formed for $Re=3000$, and $H/D=1, 2, 3$

Similarly to increasing the Reynolds number, an increase in jet to target spacing moves the center of the toroid radially outward. For the range of $H/D = 1$ to 3, at a fixed $Re = 3000$, the toroid is displaced by $1.3(r/D)$ per 1D increase of jet to target distance. All trends support the experimental work mentioned [25, 26], that is, both an increase in Reynolds number and jet to target distance, displace the toroidal vortex towards the outlet.

The recirculation patterns shown in Figures 3.5 and 3.6 clearly illustrate the entrainment of the re-circulated air into the potential core of the jet. Without the presence of a top wall, the jet entrains the ambient or surrounding air, which for a cooling application, would have a lower temperature than the heated re-circulated air. This effect is better seen for an H/D ratio of 3 (Figure 3.6), where ambient air is still being entrained.

The exchange of momentum with the surrounding air widens the jet as it discharges from the nozzle, thus, degrading the potential core. Figure 3.7 depicts the difference in velocity contours for $H/D = 6$, at Reynolds number 2000 and 6000, where the highly focused jet is seen at the higher Reynolds number:

Figure 3.7 – Velocity contours (m/s) for $H/D=6$, $Re= 2000$ and 6000

As the velocity of the jet increases, the thickness of the jet decreases, and therefore it converges more rapidly. This effect is qualitatively illustrated above, and quantitatively illustrated below, in Figure 3.8 where the thickness of the free jet is plotted for several Reynolds numbers.

Figure 3.8 – Free jet thickness with increasing Reynolds number, $H/D=6$

The jet thickness $R_{1/2}$, is defined as the distance from the symmetry line up to 50% of the peak velocity on the centerline (U_m). As it can be seen above, the jet converges more rapidly as the Reynolds number

increases. Tripling the Reynolds number from 2000 to 6000 decreases the thickness of the jet at 0.5D normal to the plate by 49.7%⁵. The significance of this localization is the effect it yields on the heat transfer rate. Due to the convergence of the jet at higher velocities, the stagnation region will shrink and affect the overall average heat transfer on the plate.

3.3.2 Stagnation Point Heat Transfer

Figure 3.9 depicts the effect of Reynolds number and jet to target spacing on the stagnation Nusselt number:

Figure 3.9 – Stagnation Nusselt number change with Reynolds number and H/D

For all values of jet to target spacing, an increase in Reynolds number consequently increases the stagnation point heat transfer. Such an increase is inevitably due to the increase of exit velocity of the jet, which inherently yields higher turbulence values. The increase of jet to target spacing has no major effect on the stagnation point. At these low Reynolds numbers and H/D range, the maximum deviation due to an increase in H/D ($H/D=1 \rightarrow H/D=6$) is 25% for the lowest Reynolds number- 2000. As the Reynolds number increases, the stagnation point becomes less sensitive to jet to target spacing. Proof of this insensitivity is the reduction of the max deviation between $H/D=1$ and $H/D=6$, to 6.6% for $Re=6000$. Albeit, even at these low Reynolds numbers, the overall pattern for an increase in jet to target spacing is a reduction of the stagnation point Nusselt number. Yet, in between the extremes of H/D

⁵ In figure 3.8, 0.5D from the plate, represents a z/D of 5.5 since the total $H/D=6$.

ratios tested, there exists a slight maxima occurring. A noticeable occurrence is at $Re=6000$ and $H/D=5$, where a sudden increase in jet to target spacing drops the stagnation point to its lowest value. Such an occurrence has been experimentally observed at higher Reynolds numbers and with larger nozzles. The work of Garimella and Rice [14] examines the behavior of stagnation point heat transfer for a confined circular jet with a nozzle 58% times the size simulated in this study. Their results for their lowest Reynolds number (8500) depict a similar trend for the highest Re in this study. That is, a relatively steady stagnation value up to $H/D=5$ and dropping thereafter for any increase in target spacing.

Given the smaller nozzle simulated in this study, and the trend previously explained from the work of Garimella and Rice [14], it is expected that the variance with jet to target spacing should be less pronounced. Gardon and Akfirat [16] observed the significance of turbulence in stagnation point heat transfer: It is dependent on the turbulence inherent in the spreading of the free jet, and the turbulence due to mixing with the entrained air, which is a function of jet to target spacing. Therefore, at values lower than the stipulated potential core length $\sim 5D$, jet to target spacing should not affect the stagnation region. Yet, it has been shown by Garimella and rice [14] and San and Shiao [56] that the sensitivity of the Nu_{sg} is also dependent on nozzle diameter. Thus, with a micro nozzle, the turbulence generated due to mixing entrainment should lessen⁶ (in the absence of crossflow), and therefore, the stagnation point heat transfer should be dominated by the inherent turbulence in the free jet and its exit profile; thereby explaining, the flat profiles shown in Figure 3.9. A similar insensitivity for jet to target spacing was found in the numerical work of Pence et al [57], who use 0.1mm nozzle with laminar Reynolds values. The transitional regime in this study, between $Re=2000$ and $Re=6000$, remains in agreement with the trends found in the mentioned experimental and numerical work.

To evaluate the stagnation point heat transfer deviation between the numerical results and experimental work, the correlations of San and Shio [56], Garimella and rice [14], and Shadlesky [58] are used for comparison. Shadlesky's [58] correlation can only be used for laminar jets, thus, it is only used for $Re=2000$. Figure 3.10 plots the predicted numerical stagnation values and the experimental correlations mentioned:

⁶ Smaller diameter yields less entrainment of ambient air

Figure 3.10 – Comparison of predicted stagnation Nusselt values and experimental data

The Reynolds numbers simulated in this study fall slightly out of the recommended range for both correlations: Garimella and rice [14] suggest $Re > 4000$, while San and Shio [56] $Re > 10000$. Nevertheless, the computational results agree well with their data. For $H/D = 3$ and $Re = 4000$, the largest deviation, 16%, is obtained from the San and Shio [56] correlation, which has a mean deviation of $\pm 6\%$. For $H/D = 6$ and $Re = 6000$, the largest deviation is given by Garimella and Rice [14], 18.5%, with an inherent mean deviation of $\pm 10\%$. For the laminar jet at $Re = 2000$ and $H/D = 1$, the numerical results deviate 3.7% from the Shadlesky [58] correlation. Overall, good concurrence is achieved for all stagnation values: For all H/D ratios and Reynolds numbers iterated, an average deviation of 16.5% and 20.5% is obtained with the Garimella and Rice [14] and San and Shio [56] correlations, respectively.

3.3.3 Local Heat Transfer

To ascertain the effect of increasing the Reynolds Number and jet to target spacing, the local Nusselt distribution is plotted for all Reynolds numbers at $H/D = 6$ and $H/D = 1$:

Figure 3.11 – Local Nusselt distribution with increasing Reynolds number at $H/D=6$ and $H/D=1$

It is evident that at a constant jet to target spacing an increase in Reynolds number shifts the Nusselt curve upwards. As the Reynolds number increases the slope of the Nusselt curve in the stagnation region slightly increases, this is the case for the larger jet to target spacing of 6. The latter cannot be said about the low H/D ratio of 1, where the curve does not get sharper with an increase in Reynolds number. A possible explanation is given by Figure 3.8, where the thickness of the jet is depicted, from this figure it can be seen that : at low Z/d (or H/D) values, the free jet is not given a chance

to expand, as such; an increase in Reynolds number does not greatly change the thickness of the jet. Thus, for all Reynolds number it yields the same thickness at a low jet to target spacing. This effect localizes the heat transfer in the stagnation region at higher Reynolds number and at high H/D ratios. Indeed, this trend is observed in experimental work [16, 20, and 46]: sharper curves at high Reynolds numbers and jet to target ratios, $H/D > 5$.

The correlation derived by Hoffman et al [19] (Eq.2.9) is plotted as well in Figure 3.11 for comparison. The disagreement between the results cannot be blamed on the numerical prediction. The correlation does not take into account the nozzle diameter, jet to target spacing, or exit velocity profile. It is a practical correlation which should be used for quick verifications. There are too many variable which affect the local heat transfer distribution under an impinging jet which would make a complete correlation unpractical (Hoffman et al [19]). Its use in this study was due to the scarce experimental data available for the nozzle diameter, and Reynolds numbers studied herein.

Figure 3.12 (and partly 3.11), depict the effect of decreasing the jet to target distance on the local Nusselt distribution:

Figure 3.12 – Local Nusselt distribution with changing jet to target spacing at $Re=6000$

Evident from the plot above is the relative insensitivity of the local Nusselt number with changing jet to target spacing; this was previously encountered with the stagnation value. The overall insensitivity of the local heat transfer at this range of H/D values has been observed experimentally in

several studies [16, 57, and 26]. Also seen in Figures 3.11 and 3.12, is the minima occurring at the stagnation point leading to inner peaks being formed in the $0.18 - 0.5 r/D$ range. The work of Gardon and Akfirat [16], and Gardon and Carbonpue [21] have observed inner peaks in the region $\sim 0.5 r/d$ from the stagnation point. These inner maxims have been observed for axisymmetric jets when $H/D < 3$, and for slot jets when $H/D < 1/2$. They have been attributed to the minimum boundary layer thickness formed at $r/D=0.5$ with the radial flow of an axisymmetric impinging jet. Saad et al [22] also observed that a jet with a fully developed profile, such as the one modeled in this study, will yield a maximum coefficient of friction near $0.5 r/D$. Yet, the numerical results fail to accurately predict the position of this maximum. For an H/D spacing of 1, the inner peaks are situated at $0.33 r/D$, rather than 0.5. The location of the maxim is not steady with a change of H/D , nor with Reynolds number. This is contrary to the findings of Gardon and Akfirat [16] where the inner peaks have a steady value near $0.5D$ regardless of Re or H/D . It is stipulated that the eddy viscosity model K-epsilon RNG is incapable of accurately capturing this phenomena.

At lower H/D ratios (<5) it is expected to see a secondary maxima occurring near $r/D=2$. This has been explained by the transition from accelerating stagnation flow to a decelerating wall jet [16, 15]. It has also been associated with boundary layer destruction through the toroidal vortices shown in Figure 3.5 [52]. None of the local Nusselt distributions in this study show any indication of a secondary maxim occurring, which is in accordance with the low Reynolds numbers and the top confinement wall. Gardon and Akfirat [16] find these maxims to occur for axisymmetric jets at Reynolds number higher than 2500, yet the work of Garimella and rice [14] show no secondary maxims below $Re = 6000$. The key difference between the experimental works mentioned is the top confinement wall present for the latter. Clearly, the recirculation zone formed (which is due to the top confinement wall) retards or accelerates the boundary layer transition, which gives rise to the secondary maxims encountered for $H/D < 5$. How it specifically affects the boundary layer transition would require a larger control volume to be modeled, and it is therefore out of the scope of this analysis. Overall, for the range of parameters simulated in this study, the turbulence model K-epsilon RNG does well in not predicting any secondary maxims near $r/D=2$.

The trends discussed can all be seen in the figures below. Figure 3.13 illustrates the Nusselt number contours on the plate with changing Reynolds number at $H/D=1$. Figure 3.14 depicts the Nusselt number contours with changing jet to target spacing a constant Reynolds number of 6000:

Figure 3.13 – Nusselt number contours on target plate, effect of increasing RE, H/D=1

Figure 3.14 – Nusselt number contours on target plate, effect of increasing H/D , $Re=3000$

The general insensitivity to jet to target spacing can be seen in Figure 3.14, where the Nusselt footprint does not vary significantly across the different H/d ratios, which was quantitatively shown in Figure 3.12. Conversely, the enhancement of heat transfer can be clearly seen with increasing Reynolds number, particularly in the stagnation region.

3.3.4 Average Heat Transfer

To obtain the effect of increasing Reynolds number on the average heat transfer, a constant H/D ratio of 6 is used- Figure 3.15 illustrates the trend. The correlations used for comparison are Martin [15], and , although slightly outside the recommended range of use, Huang and el-Genk [18], and Hoffman et al [19], the latter having applicability at $Re > 6000$ and $Re > 14000$, respectively.

Figure 3.15 – Average Nusselt number with increasing Reynolds number, $H/D=6$

As expected, at a fixed jet to target spacing, the average heat transfer on the plate increases with increasing Reynolds number. The deviations with the correlations are shown both on Figure 3.15 and Table 3.5.

Table 3.5 – Average Nusselt number deviation with experimental correlations, H/D=6

		Deviation %		
		Martin	Huang	Hoffman
Re	2000	-2.89	-27.97	-21.54
	3000	-3.04	-22.20	-15.63
	4000	-1.05	-19.81	-13.31
	5000	0.22	-17.82	-11.37
	6000	1.80	-16.70	-10.32

Excellent agreement is obtained with the work of martin [15], and good agreement is obtained for Re=6000 for Huang and el-Genk [18], and Hoffman et al [19]. The poor agreement at the lower Reynolds numbers with the latter correlations is due to the recommended Re applicability. This is clearly seen with a decreasing deviation as the Reynolds number increases.

At a fixed target spacing, the average Nusselt number on the target plate has been shown to correlate with the Reynolds number like so [20]:

$$\overline{Nu} \propto Re^b \quad \therefore \overline{Nu} = ARe^b \quad (3.5)$$

It is possible to use the values predicted by the numerical analysis to find the exponent “b” in equation 3.5. This task yields a value of 0.628. A similar analysis by Goldstein and Behbahani [20] with larger Reynolds numbers (>35000) and a nozzle 25 times larger obtains an exponent of 0.6 for the variance of average heat transfer with Reynolds number. What this entails is the correct prediction of increasing average heat transfer with increasing RE.

The effect of changing the jet to target spacing on the average heat transfer is illustrated in Figure 3.16.

Figure 3.16 – Average Nusselt number with increasing jet to target spacing, Re=6000 and Re=3000

Although it is relatively small, the overall effect is that of a decreasing heat transfer with an increase in jet to target spacing. An 8% decrease in average heat transfer is obtained when increasing the jet to target spacing from 1 to 6 - Martin's [15] correlation predicts 10%. Between the extremes of $H/D = 1$ and 6, the numerical results indicate slight increases which none of the correlations predict. Specifically, the heat transfer increases from $H/D = 1$ to $H/D = 2$ by 8 % at $Re = 3000$, and 5% at $Re = 6000$. The trend seems to indicate that this effect will disappear at higher Reynolds numbers. Nevertheless, the overall deviations with the proven correlations are small enough to concur that the turbulence model achieves good agreement with experimental data.

3.4 Conclusions

A single axisymmetric impinging micro-jet was modeled, meshed, and solved numerically. The diameter of the jet was fixed at 0.5mm, the Reynolds numbers ranged from 2000 to 6000, and the jet to target spacing from 1D to 6D.

Several turbulence models, the standard K-epsilon, K-epsilon RNG, K ω , and K ω -SST were compared to referenced experimental data. It was found that the K-epsilon RNG best suited this impingement model as it obtained the smallest deviation for the local and average heat transfer values.

Streamlines for the impingement flow field were extracted and toroidal vortices near the impinging jet were obtained in the recirculation pattern. The vortices moved radially outward as the

Reynolds number and jet to target spacing increased. The same trend has been observed experimentally for macro jets and at higher Reynolds numbers.

Heat transfer in the stagnation region indicated less sensitivity towards jet to target spacing than larger macro-jets in experimental work. Comparison to experimental correlations yielded good agreement, ranging from 3% to a net deviation of 14%. The local Nusselt distribution remained relatively insensitive to jet to target spacing, but sharper curves were obtained with higher Reynolds number and H/D ratios. The sharpness was attributed to the increasing thickness of the jet at higher z/D values. The inner peak near $r/D=0.5$ which has been experimentally observed for axisymmetric jets at low H/D ratios was obtained, but the turbulence model was incapable of accurately capturing its location. No secondary maxims were predicted, which was in accordance for the range of parameters modeled in this study. The overall Average heat transfer on the plate increased with increasing Reynolds number, and decreased slightly, 8%, when the jet to target spacing is increased from $1D$ to $6D$. Good concurrence was achieved for the average heat transfer in comparison to experimental correlations.

The objective of this chapter was to gain confidence in the commercial package Fluent in predicting the flow and heat transfer physics of a single circular impinging micro- jet. Now that a single jet has been modeled, understood and validated, the analysis can continue to the more complicated setup of a confined impinging in-line jet array.

4.0 HPT Vane – Validation Model

Single jet impingement cooling is seldom used by itself. Certain instances are known where a small number of slot jets are used, for example, in the leading edge of a rotor, or in industrial drying. In HPT vanes, as was depicted in the introduction, the number of jets is typically greater than 100. The behavior of a single jet was modeled in the previous chapter, with the results indicating no significant effect on the average heat transfer with changing jet to target spacing. In order to obtain a realistic analysis of an HPT vane, an accurate geometry and impingement array must be modeled. Only then is it possible to assess the effect of using micro-jets to cool the airfoil.

4.1 Vane Geometry

The chosen vane is the previously mentioned 2nd stage vane of the GE E³ engine (Halila et al [8]). The convection only cooling system for this vane makes it an excellent model for the study of impingement cooling under a realistic geometry. The vanes are fed from the 7th stage compressor stator bleed, where 4 pipes route the air towards 8 casing inlet ports. A total of 1.85% of the core flow is used to cool the 2nd stage vanes, as oppose to 9.24% for the 1st stage. The vane utilizes a single impingement insert with an array of jets of 0.51mm diameter. The insert utilizes preferential cooling to account for the expected higher heat load at 95% span, but an average spanwise jet spacing of 4D is used. The spacing streamwise varies linearly with a higher concentration in the leading edge. Figure 4.1 depicts the two dimensional profile of the 2nd stage vane. Leading edge cooling is used, and all the spent air is forced to exit on the pressure side bleed slots, which serve to cool the trailing edge of the vane.

Figure 4.1 – 2nd stage HPT vane cross section [8]

The 2nd stage vane uses a rene150 airfoil profile:

Figure 4.2 – rene150 airfoil profile for 2nd stage vane [8]

Using the profile above and a scaled image of the full cross section, it was possible to extract the thicknesses of each surface: airfoil thickness, impingement cavity thickness. Assuming that the profile remained approximately the same for all inner surfaces (verified by the constant spacing all around), a model was constructed in SolidWorks.

The equations dictating the airfoil shape were not known; therefore, tracing in SolidWorks had to be used. Smooth curves were created, and the dimensions were verified. Inner cavities were then created by cutting the airfoil block at appropriate thicknesses. Figure 4.3 depicts the general model with and without a left face, respectively:

Figure 4.3 - Completed vane model

No impingement holes are shown in Figure 4.3 because their position varied depending on the actual or validation model⁷. The model shown above represents an exact replica of a spanwise segment of the 2nd stage vane shown in Figure 4.1. The span length was chosen to accommodate 3 columns of jets of $D=0.51\text{mm}$, and spacing of $4D$, with $2D$ from the edge walls. At least 3 columns of jets were required to obtain results for a middle jet, which best represents an arbitrary jet not affected by the end walls of the airfoil.

The flow occurring in such a model replicates a segment of the 2nd stage vane, and as it can be seen, it is fully three dimensional. That is, the flow enters the impingement chamber in the z -direction, the jets discharge perpendicularly into the inner surface, and subsequently it is forced to turn streamwise towards the trailing edge. As mentioned in chapter 2, this is usually not the case with experimental or numerical studies, where the direction of the flow into the plenum is aligned with the jet discharge itself. Figure 4.4 best depicts the situation:

⁷ Refer to 4.2 or 4.3 for impingement hole locations

Figure 4.4 – General flow direction through vane

The nature of this model prevents symmetry to be used. Since the air enters the inlet (green face) from one side, the model is naturally asymmetric. Because of this, subtle differences are expected from neighbored spanwise jets, which usually is not the case in most experimental studies.

The following table summarizes the key characteristics of the default vane model depicted above:

Table 4.1 – Geometric parameters for base vane

Suction surface curve length	62.81 mm
Pressure Surface curve length	51.4 mm
Span	6.12 mm (12D)
Jet hole diameter	0.51 mm
Airfoil Thickness	1.35 mm
Impingement plate thickness	0.54 mm
L/D (hole length / Diameter)	1.05
H/D	1.588

4.2 Necessity and Strategy for Validation Vane

As mentioned in chapter 2, there is no experimental or numerical work that has analyzed a full array within an airfoil profile. Therefore, to ascertain the validity and capability of the commercial CFD package, a simplified vane model must be validated. Experimental work, see 2.2, models the mid-chord section of a vane as a flat plate. Forcing the air to exit in only one direction (Figure 2.4) the experiment is capable of replicating most of the flow features encountered in a vane. Before the results for the full vane model can be trusted, an assessment must be made for an array which has been correlated experimentally. The strategy is as follows:

- Using the same geometry as the full vane (Figure 4.4), strategically position the impingement holes in the flattest part of the airfoil.
- Create an impingement array with geometric parameters which fall within proven experimental correlations.
- Assess the capability of the different turbulence models and the overall CFD package to replicate flow features, and most importantly the heat transfer.
- Choose the turbulence model which yields the closest results, and continue to analyze the full vane with actual impingement jet locations.

The overall objective of the validation model is to gain confidence in the computational results of a fully three dimensional array which includes the effect of spent air. Once the strengths and weaknesses of the simulation are known, the use of micro-jets in an HPT vane can be tested.

4.3 Modeling and Simplification

Figure 4.5 depicts the plenum chamber for the validation model:

Figure 4.5 – plenum chamber for validation vane ($H/D=1.58$)

The strategic location of the impinging holes is clearly seen: located towards the trailing edge on the suction surface, the jets should experience no significant curvature. The impingement array is square, that is, a pitch(S) of $4D$ is used between the jets spanwise and streamwise. Near wall jets are located $2D$ from the airfoil wall, which replicates the work Florschuetz [31]. The geometric parameters of the vane remain the same as the Table 4.1. A total of 15 jets are protruded on the plenum wall, with 5 rows of 3. The holes themselves are simple cylindrical holes of $D=0.51\text{mm}$, located at 0.8058mm from the inner surface of the airfoil, yielding a jet to target spacing of 1.58 .

The SolidWorks model cannot be meshed as it is shown in Figures 4.3 and 4.5. The model was imported via parasolid format to ANSYS ICEM, where it was trimmed for meshing. The analysis does not take into account conduction through the airfoil walls, and therefore, all wall thicknesses are redundant. The following figure illustrates the clean and trimmed cross section of the model with the various volumes and surfaces:

Figure 4.6 – Preliminary validation vane depicting main surfaces and volumes

The above figure illustrates that the flow after impinging is free to move towards the leading edge and trailing edge. The final model will include impinging jets all around the plenum, therefore, the flow is forced to exit towards the trailing edge. In order to be as accurate as possible, and to replicate the experimental work discussed in chapter 2, a cut was introduced in the mid chord region. The introduction of a wall located 2D from the closest leading edge jet, forces the air to exit at the outlet, thereby duplicating the same boundary conditions found in the experimental work cited in chapter 2 [28-33]

Figure 4.7 depicts the validation model before meshing:

Figure 4.7 – Validation model before meshing

The inlet, colored in blue, is removed on the right so the plenum cavity can be seen. Part of the process of cleaning the model for meshing purposes involves removing any wall thicknesses. Comparisons of the SolidWorks model (Figure 4.4) and the above completed model, illustrates that the thicknesses have been voided out, and thus, only volumes surrounded by the walls are meshed and solved. A closer look at the cross section and inner cavities is shown below:

Figure 4.8 – Validation model cross section and inner impingement cavity

In the above figure which depicts the flow direction, the main impingement target wall was removed. It is then possible to see the impingement volume region. Not seen in the figure, is the segmented target wall where the jets strike. In order to obtain the average heat transfer values under

the jets, the target wall was segmented into patches representing the effect of each individual jet. The reader is referred to appendix A4 for a detailed figure.

4.3.1 Meshing

Meshing the vane was done using a hybrid approach: an unstructured tetrahedral mesh is used in all surfaces and volumes, with a structured boundary layer mesh on the target impingement walls. The boundary layer mesh was a necessity in order to assure a certain y^+ value (see chapter 4, section 4.3). The same meshing approach was used by the numerical work of Kumar and Prasad [49], which modeled leading edge impingement cooling for a row of jets.

ANSYS ICEM tetrahedral mesh tool can be used to set different mesh element scales for different volumes and surfaces. The model is split into three different fluid volumes: the plenum, the impingement region, and the individual jets. Different element sizes were assigned to each in order to achieve a required resolution depending on the importance of the location itself. The plenum had the largest elements, with the jets and the impingement region set at approximately the same level.

The boundary layer mesh was created using the Prism function of ANSYS ICEM. The required specifications are: the height of the closest wall node, the number of layers, and the ratio for which the distance increases (a logarithmic ratio was used).

Depending on the model meshed, $H/D=1.58$ or $H/D=3$, different scales were used, but the approach remained the same and mesh sensitivity was always verified. A look at the intermediary mesh for the $H/D=1.58$ model is depicted below (surface mesh only):

Figure 4.9 – Validation model $H/d=1.58$, intermediary mesh

To capture near wall behavior, a boundary layer mesh is used only above the target impingement surface:

Figure 4.10 – Validation model $H/D=1.58$, boundary layer mesh

The jet exits and inlets are meshed using very fine elements to capture the velocity distribution. The jet walls use a larger element size (internal volume mesh not shown for clarity):

Figure 4.11 – Validation model $H/D=1.58$, jet mesh

4.3.2 Boundary Conditions and Numerical Setup

The flow through the HPT vane is driven by the pressure difference imposed at the inlet and outlet. Figure 4.12 depicts the defined boundary conditions for the validation model:

Figure 4.12 – Validation model $H/D=1.58$, boundary conditions

A gauge pressure is specified at the inlet surface, the pressure outlet is left at ambient pressure. The delta pressure, or pressure drop, drives the flow through the jets and towards the pressure outlet. The choice for a pressure inlet boundary condition was due to the known pressures used in the actual vane (Halila et al [8]). Nevertheless, using a velocity inlet and an “outlet” yield the same results (if the inlet velocity for that specific inlet pressure is matched). The inlets and exits of the individual jets themselves are defined as interiors in order to let the flow go through. Inner/outer plenum and side

walls are all defined as adiabatic walls. Realistically, the side walls would experience a heat flux from the hot gases, but to better replicate experimental work, they are left adiabatic.

Initially, the outer airfoil surface was imposed with a higher temperature than the impinging air. In realistic conditions, the inner airfoil surface has a temperature near 950°C, and the bleed air comes in at approximately 600°C (Halila et al [8]). Of main concern to this study is the enhancement of the heat transfer coefficient, which is independent of the temperature. Difficulties were experienced converging at high Reynolds numbers when the airfoil surface had a higher temperature than the impinging air. The solution to this problem is to reverse the temperatures, heat the impinging air, and fix a low ambient temperature on the airfoil surface. Such a method is usually used in experimental work, since it is easier to heat the air itself, and, the heat transfer enhancement is still captured. For this study, the airfoil surface was given a constant temperature boundary condition of 300°K, and the impinging air was set to 500°K.

Due to the small diameter simulated in this vane, 0.51mm, high velocities are required to achieve the Reynolds numbers which experimental correlations are typically valid for, $Re > 5000$. All simulations used constant properties (μ , ρ , k , c_p), and to avoid compressibility effects the Mach number had to be kept below 0.5. The high temperature of the air assisted in increasing the speed of sound, but it was not enough to achieve the highest desired Reynolds number, 15000, and keep the velocities below compressibility effects⁸. Therefore, the density was set to a realistic HPT vane value so as to increase Reynolds numbers. In Fluent, the density was set to "incompressible-ideal gas", which utilizes the ideal gas law equation and the specified ambient pressure to calculate the density. Ambient pressure was set to $P = 1.435$ MPa, which for 500°K air yields a density of 10 kg/m^3 . In typical engine conditions, the bleed air entering the HPT vanes will have a density near 12 kg/m^3 (Halila et al [8]). Table 4.2 summarizes the boundary conditions and the fluid properties for the validation simulation:

⁸ Experimental work typically uses larger diameter nozzles $> 5 \text{ mm}$, which makes it possible to achieve higher Reynolds numbers [28,33,37].

Table 4.2 – Validation model boundary conditions & property values

Surface or Property	Condition set	Value
Inlet	Pressure-Inlet	Gauge P – varied depending on case
Outlet	Pressure - Outlet	Ambient Pressure
Plenum and Side walls	Adiabatic Wall	N/A
Airfoil surface	Constant temperature	300°K
Impinging Air	Constant temperature	500°K
Ambient/Operating Pressure	Constant	1.435 MPa
Density (ρ)	Incompressible/Ideal Gas	10 kg/m ³
Specific heat (C_p)	Constant (at 500°K)	1030.305 J/kg°K
Thermal Conductivity (k)	Constant (at 500°K)	0.040284 W/m°K
Viscosity (μ)	Constant (at 500°K)	2.6375 x 10 ⁻⁵ kg/ms

All solutions were considered converged when all normalized residuals (continuity, x-y-z velocities, and turbulence variables) achieved a level of 10^{-4} , except for energy which required 10^{-6} . These levels were considered appropriate due to the previous results obtained with a single jet (chapter 3), and other numerical work [49, 51]. The implicit pressure based solver is used, with the pressure velocity coupling scheme set to SIMPLE⁹. The model has no *significant* curvature, nor high values of natural convection and body forces, therefore, the discretization/interpolation of pressure is left as STANDARD (as suggested by Fluent [59]). Momentum, energy, and turbulence variables are discretized using a 2nd order upwind method, as it should yield more accurate results for a hybrid grid (Fluent [59]). Under relaxation was used in order to achieve convergence for certain cases.

4.4 Mesh Sensitivity

A mesh refinement study was performed in order to assure the results were independent from the grid. Three successive grids were created and the Grid Convergence Index (Roache [54, 55]) was used to verify that they lied in the asymptotic range. The GCI index is typically used for structured hexahedral grids, but the method is valid for hybrid tetrahedral grids such as the one used in this study [55]). The refinement for each grid was performed by shrinking the element size at respective locations, thereby increasing the number of nodes. This type of refinement is not fully structured, that is, where 1 tetrahedral element is split into 4 new elements, and hence, a quantifiable refinement factor “ r ” cannot be obtained by using the decreasing element size. Roache suggest using the following definition of effective refinement factor:

⁹ Several models were iterated using SIMPLER, and no difference was seen in convergence time.

$$R_{effective} = \left(\frac{N_F}{N_C}\right)^{1/D} \quad (4.1)$$

Where N_F and N_C is the total number of elements in the fine, and coarse grids, respectively. D is the dimensionality of the problem, which has been chosen as the diameter of the jet for this study. Reporting the GCI index in a refinement study is preferable to simply reporting relative error, but it does not have the same firm basis as it does for a structured hexahedral grid [54, 55]. Therefore, to compliment the GCI index analysis, the crossflow distribution, average and local heat transfer values are compared across the grids.

Average heat transfer values on the impingement target plate are used to calculate the GCI index¹⁰. The following tables summarize the results:

Table 4.3 – Average Heat transfer for three validation grids

Grid	$h_{avg}(W/m^2 \cdot K)$	% dev
Coarse	5207.99	
Medium	5426.21	4.021551
Fine	5479.20	0.967066

Table 4.4 – GCI calculation

Effective refinement factor	1.59
Zero Spacing h_{avg} Value	5496.19
GCI coarse %	1.612
GCI Fine %	0.985

From Table 4.3 and 4.4 it is evident that further refinement would not significantly increase the accuracy. A GCI index of 0.985% yields the maximum discretization error for the prediction of the average heat transfer value under the jets. As mentioned, due to the unstructured nature of the grid, the GCI index could underestimate or overestimate such a value. Furthermore, it does not take into account local discretization errors. For that reason, the crossflow ratio¹¹ G_o/G_j is also obtained for all three grids. Figure 4.13 depicts the crossflow buildup across the array:

¹⁰ The reader is referred to [55] for specific details on GCI calculation

¹¹ See 2.2 for explanation

Figure 4.13 – G_c/G_j ratio for three grids

It can be seen in the above figure that all three grids yield the exact same crossflow to jet mass velocity ratio distribution across the array. The low deviation between the grids for this ratio is to be expected, as it deals with the overall continuity of the array, and not the near wall behavior which is the most challenging region to capture.

The average spanwise heat transfer coefficient is also compared:

Figure 4.14 – Average spanwise heat transfer coefficient for three grids

A clear difference is seen between the coarse and medium grid, but no significant difference is seen between the medium and fine grid. Maximum deviation tends to occur in the 4th row of jets at $x/X_n=3.5$.

Local heat transfer values under the 1st column of jets (left column)¹² are extracted for all three grids. Figure 4.15 depicts the distribution:

Figure 4.15 – Local heat transfer distribution for all three grids

It is clear from looking at Figure 4.15 that the local heat transfer values do not change drastically between the three grids. Regions of deviation include stagnation points as expected, and downstream from the 4th jet, where the crossflow is high.

The overall average heat transfer on the airfoil surface differs by only 0.96% between the medium and fine grid. In terms of computational requirements, the fine grid requires nearly double the total time to reach convergence than the medium grid. Given the subtle difference between the grids, both in average and local values, the medium grid resolution is chosen to continue validating the vane. The maximum y^+ value with the medium grid is 4.5 (near the trailing exit of the array); Average y^+ value is below 1. An iterative procedure was followed to assure that the y^+ value remained below 5 for all cases. If a high y^+ value is obtained, the starting node near the wall is modified from the prism layers in ANSYS ICEM.

¹² For a detailed explanation of local heat transfer values and their extraction method see 5.4.3 and appendix A9

4.5 Results & Validation

In chapter 2, section 3, a review of the numerical work for impinging jets is summarized. The assessment revealed that three RANS turbulence models are preferred and have been validated for impinging jets: v^2 -f, K ω -SST, and K- ϵ RNG. There is no debate in current literature that the v^2 -f model yields the most accurate results, but unfortunately, no license was available for the turbulence model at the time of this study. Given the time span of this project, there was also no possibility to use non RANS models, such as Direct Numerical Simulation (DNS), Detached Eddy Simulation (DES), and Large Eddy Simulation (LES). Therefore, to assess the predictions for the validation model, two turbulence models are compared: K ω -SST and K- ϵ RNG. The reader is referred to chapter 2, section 2.3 for a more in depth look at how these turbulence models have been used for impinging jets.

4.5.1 Crossflow to Jet Mass Velocity Ratio

The crossflow mass velocity (G_c) to jet mass velocity (G_j) ratio defines the crossflow buildup across the array. The air is forced to exit in only one direction, which means that the mass flow will increase towards the exit due to the downstream jets introducing more air. The distribution of this ratio across the array is crucial, since, the crossflow diminishes the performance of the jets. Upstream jets will experience nearly no crossflow, but the last (5th row) of jets will be swept by the entire accumulation of spent air.

The G_c/G_j ratio is resolved streamwise, which entails averaging the mass velocity of the jets spanwise. The crossflow mass velocity G_c (kg/m²s) is defined as:

$$G_c = \frac{\dot{m}_c}{A_c} \quad (4.2)$$

The crossflow mass flow rate ($\dot{m}_c$) is the average mass flow between half a pitch upstream and half a pitch downstream from the jet. A_c is the cross-sectional area of the channel (H x Span of the vane). The reader is referred to appendix A1 for a diagram illustrating the above. The jet mass velocity G_j (kg/m²s) is defined as:

$$G_j = \frac{\dot{m}_j}{A_j} \quad (4.3)$$

The jet mass flow rate ($\dot{m}_j$) is the average mass flow rate of the 3 spanwise jets for that specific row. A_j is the cross sectional exit area of the jet (πr_j^2).

G_c/G_j is a function of the array geometry (see 2.2); therefore, an increase of Reynolds number does not change the cross flow distribution. To ascertain the results, two different models at different jet to target ratios are compared to the Florschuetz [31] correlation (Eq.2.12): $H/D=1.58$ at $Re_{avg}=5000$ ¹³, and $H/D=3$ at $Re_{avg}=15000$. Figure 4.16 and 4.17 depict the crossflow distribution for the mentioned cases:

Figure 4.16 – Crossflow to jet mass velocity ratio distribution for $H/D=1.58$, $Re_{avg}=5000$

Figure 4.17 – Cross flow to jet mass velocity ratio distribution for $H/D=3$, $Re_{avg}=15000$

¹³ Re_{avg} is the effective average Reynolds number of all the jets in the array.

As it can be seen, both turbulence models capture the crossflow distribution very accurately at both H/D ratios. The Florschuetz correlation (Eq.2.12) depends on the average discharge coefficient of the array, which both turbulence models agree on: 0.75 for $H/D=1.58$ at $Re=5000$, and 0.77 for $H/D=3$ at $Re=15000$. A more uniform distribution is achieved with the higher jet to target spacing of $3D$, this is seen with the slight curvature in Figure 4.16. Both turbulence models are capable of capturing the overall decrease in crossflow with an increase in jet to target spacing. Maximum crossflow occurs at the last row of jets, where an increase of 42.7% is seen in the G_c/G_j when the jet to target spacing is nearly halved ($H/D=3 \rightarrow H/D=1.58$).

The crossflow mass velocity ratio is dependent on the jet exit and crossflow velocities. Therefore, the accurate prediction of this ratio entails an accurate prediction of the velocities for this array. A fair amount of confidence can be given to both turbulence models for predicting the velocity distribution for full vane geometry.

4.5.2 Reynolds Number Distribution & Stagnation Point Heat Transfer

The single jet analysis revealed that the maximum heat transfer, stagnation point, is usually found directly below the jet ($r/D < 1$). With an array, it is expected that the crossflow will move the stagnation points downstream from under the jets. Similar to a single jet, the stagnation point is very sensitive to many variables and has proven quite difficult to correlate successfully for all impinging arrays. A direct comparison of stagnation point values would require replicating the exact experimental conditions and jet diameter¹⁴. Therefore, the objective of this section is not to compare the stagnation point directly to experimental work, but rather, to verify that the trends are similar. More specifically, experimental work (Van Treuren [28,29]) has found: Increasing the Re_{avg} always results in an increase in stagnation point Nu for all H/D values; Local Reynolds values increase across the array due to increasing crossflow velocity, which in turn *can* increase the stagnation point heat transfer. The latter observation mentioned is not always the case. The work of Gao [38] has found that the stagnation points do not increase monotonically across the array, even if the local Reynolds values do increase.

The local Reynolds number can be averaged spanwise and plotted streamwise for each row, revealing the Reynolds distribution. Both turbulence models predict the same local Reynolds distribution. Maximum deviation occurs for $H/D=1.58$, $Re_{avg}=5000$, at the third jet row, 1.5%. As

¹⁴ It was shown for a single jet that the stagnation Nusselt number has a dependence on diameter (Garimella and Nenaydykh [41]); no experimental work has correlated the stagnation point values for an array with jet diameter.

expected, the Reynolds number increases across the array. Figure 4.18 illustrates the distribution across the array for several cases (only the results from the RNG model are shown due to the extremely small deviation for K ω -SST):

Figure 4.18 – Reynolds number distribution for $H/D=1.58, 3$ at $Re_{Avg}=5K, 15K$

Several trends are seen on the above figure. Comparing at the same jet to target ratio, the overall distribution remains geometrically similar, that is, the shape is not altered for an increase in average Reynolds number. For a jet to target spacing of $H/D=1.58$, the increase in Reynolds number from the 1st row to the 5th row is 19.8%, at both $Re_{avg}=5K$ and $15K$. At the larger jet to target ratio of $3D$, the Reynolds number is relatively flat compared to its lower H/D counterpart; at both average Reynolds numbers the local Reynolds number increases only by 7.6%. The experimental work of Florschuetz [31] and Van Treuren [28] has found the same trend of increasing uniformity with increasing H/D for square arrays such as the one modeled here. The flat profile for high H/D values has been attributed to a more uniform pressure distribution due to the larger channel area and thus lower crossflow velocities. The numerical results agree with the experimental trends, indicating a good agreement hydro-dynamically¹⁵.

With a single jet, an increasing velocity or Reynolds number yields a higher stagnation number. If the crossflow is not significant in an array, it can then be expected that the stagnation point heat transfer would increase across the array. The difficulty with the presence of a significant crossflow is

¹⁵ Note these values are indirectly verified through the crossflow to jet mass velocity ratio, nevertheless, local Re values needed to be verified for increasing uniformity with a higher H/D ratio.

that the effective jet exit velocity or Reynolds number is reduced as it encounters the crossflow. The upstream jets do not have this problem because the crossflow is low at their location, and therefore, the jet exit velocity is a good indication of stagnation point heat transfer. Defining the Nusselt number as:

$$Nu = \frac{hD}{k} \quad (4.4)$$

The maximum Nusselt number, averaged spanwise, is plotted for each row to assess how the stagnation point increases/decreases across the array: $H/D=1.58$, Figure 4.19.

Figure 4.19 – Stagnation values across the array for $H/D=1.58$ at $Re_{avg} = 5K, 15K$

Figure 4.20 depicts the distribution for a jet to target spacing of $3D$:

Figure 4.20 – Stagnation values across the array for $H/d=3$ at $Re_{avg}=5K, 15K$

For all Reynolds numbers and H/D ratios, the Kw SST model yields higher stagnation values than the RNG. The increasing/decreasing trends is the same across all cases, except for $H/D=1.58$ at $Re_{Avg}=15K$, 5th row, where RNG predicts a decrease from the 4th row and Kw SST obtains a slight increase.

At the $H/D=1.58$ the stagnation number slightly increases due to the higher exit velocities, as seen in Figure 4.18, but leveling off rather quickly near the 4th and 5th row; a consequence of the increasing crossflow velocities which reduce the effective penetration of the impinging jet. A figure depicting the velocity vectors for the middle column of jets at the 2nd and 5th row illustrates this situation ($H/D=1.58$ at $Re_{Avg}=5000$):

Figure 4.21 – Velocity vectors (m/s) for middle jets at rows 2 and 5, $H/D=1.58$, $Re_{Avg}=5000$, RNG

As it can be seen, although row 5 has a higher exit velocity than row 2, its normal velocity component striking the impingement surface is effectively reduced due to the crossflow. The sensitive balance between the crossflow and jet velocities determines the stagnation point heat transfer. The length of the jet potential core is usually $\sim 5D$ (see chapter 3), and therefore, doubling the jet to target spacing to an H/D value of 3, should not greatly affect the stagnation values; this was the case for a single jet. Comparing at constant Reynolds number of 15K with a changing H/D ratio, Figure 4.19 and 4.20, the average stagnation value across the array decreases by 12% (for Kw SST, 15% for RNG) when the jet to target spacing increases to 3D. A correlation given on Van Treuren [28] which accounts for crossflow can be used to compare the predicted average stagnation heat transfer drop with increasing H/D . With the crossflow distribution for this array, an increase from $H/D=1.58$ to 3, yields 0.5% drop in the average stagnation value. Although no direct comparison can be made regarding the actual stagnation values, comparing the changes with H/D ratio is possible. Both turbulence models over-predict the percentage drop by a maximum of 14.5%, where the Kw SST model yields the better agreement. It is possible that the stagnation values are dependent on too many variables which make a correlation unpractical. Van Treuren [28] uses the local Reynolds number and G_j/G_j ratio to obtain the stagnation value, this yield a stagnation NU which increases across the array. Yet, the work of Gao [38] who uses a square array of similar parameters as this numerical setup & Van Treuren [28], obtains a

non-monotonic variation of stagnation point Nusselt number. Therefore, scrutiny must be used when the stagnation values predicted by both turbulence models are used.

4.5.3 Streamwise Resolved Heat Transfer

Of great interest to a gas turbine blade designer is how the average spanwise heat transfer is distributed across the array. It is rare that local Nusselt number values under impinging arrays are correlated, as they are extremely sensitive to a variety of parameters. On that note, to validate the heat transfer results of both turbulence models, the previous cases that have been analyzed, $H/D=1.58$ & 3 at $Re_{Avg}=5K$ & 15K, will be compared to established correlations yielding the streamwise resolved Nusselt number. This includes the experimental work of Florschuetz [31], Bunker [33], and Katti and Prabhu [34]. The reader is referred to chapter 2, section 2.2 for an explanation of their work and specific correlations.

Figure 4.22 plots the average streamwise Nusselt number for $H/D=1.58$ at $Re_{Avg}=5000$:

Figure 4.22 – Streamwise resolved NU for $H/D=1.58$ at $Re_{Avg}=5000$

Both turbulence models predict a decreasing average Nusselt number across the array. The heat transfer is expected to decrease due to the crossflow diminishing the efficiency of the impinging jet. Kw SST has in average a 7% higher value than the one predicted by RNG. The decreasing trend in Nusselt number is more obvious in the Bunker correlation than Florschuetz, which yields a slight increase near the end of the array. This has been attributed to the forced convection of the channel flow behavior at lower jet to target ratios (the work of Van Treuren [28] has as well observed this phenomenon). The

numerical results therefore, resemble, in terms of trend across the array, the experimental work of Bunker who has a continuous decrease across the array for this jet to target ratio (and crossflow ratio). Yet, the overall deviation is lower when comparing with Florschuetz¹⁶: an average deviation of 6.2%, and 0.3%, is obtained for the RNG, and K ω SST models, respectively. For the Bunker [33] correlation, the RNG and K ω SST models have an average deviation of 19.8%, 14.7%, respectively.

An increase in Reynolds number increases the heat transfer on the target plate. Figure 4.23 depicts the Nusselt number profiles on the airfoil surface for $H/D=1.58$ at $Re_{Avg}=5k$ and $15K$ (K ω SST). The same color map has been used for both cases in order to better illustrate the enhancement with increasing Reynolds number:

Figure 4.23 – Nusselt number contours for $H/D=1.58$ at $Re_{avg} = 5K$ and $15K$, K ω SST

The enhancement of heat transfer when tripling the Reynolds number is clearly obvious in the above figure. Not only is the stagnation region enhanced (red regions) but the overall footprint of the jets are larger. Regions in between the jets also benefit from increasing velocities, which is seen with the more pronounced green surroundings around the jets. As was discussed earlier, the crossflow sweeps the downstream jets, and this changes their Nusselt footprint. Early rows have a more circular center

¹⁶ The Florschuetz correlation has a mean deviation of 5.6%

footprint¹⁷, which is typical of a single jet (see Figures 3.13 and 3.14). As the crossflow increases through the array, upstream flow strikes the potential core of a downstream jet and splits into two horseshoe vortices, which go around the jet thereby changing its cross-sectional shape to a “drop” like figure. The same trends of swept Nusselt number footprint, and increased heat transfer for in-between regions are seen in the work of Gao [38].

The streamwise resolved Nusselt numbers are also extracted for this case, and can be compared to the experimental correlations, Figure 4.14:

Figure 4.24 – Streamwise resolved Nu for $H/D=1.58$ at $Re_{Avg}=15K$

Comparing to $Re_{Avg}=5000$, the average heat transfer value is 55% higher. The trend predicted remains the same: streamwise Nu decreases across the array. At this Reynolds number, both correlations predict a leveling off of Nusselt number from 4th to the 5th row, which is not captured perfectly by either of the turbulence models. Yet, the deviation between the correlations and the numerical predictions is significantly better than at $Re_{Avg}=5000$. The overall average deviation from the Florschuetz correlation is 3.36%, and 7%, for the RNG and Kw SST models, respectively. With the Bunker correlation, RNG deviates 9.2%, while Kw SST only deviates 0.6%. An overall good agreement is obtained for a jet to target spacing of $1.58D$, at a low Reynolds number of 5000 and high of 15000.

¹⁷ Semi-squared footprints are also seen in figure 4.23. This is due to the wall and surrounding jets. As the Reynolds number increases, the interaction between neighbored jets and walls becomes more pronounced, leading to the footprint to conform to the square array geometry. At a larger jet to jet spacing and lower RE values, the jets retain their single nature, thereby generating a typical circular footprint.

For a jet to target spacing of 3D, the Katti and Prabhu [34] correlation is also used for comparison as it was not valid for $H/D=1.58$. Figure 4.25 plots the Nusselt number for $H/D=3$ at $Re_{Avg}=5000$:

Figure 4.25 – Streamwise resolved Nu for $H/D=3$ at $Re_{Avg}=5000$

Both turbulence models predict an overall decrease of Nusselt number across the array, which agrees with the correlations of Florschuetz [31] and Bunker [33]. The correlation of Katti and Prabhu predicts a rather flat distribution across the array but fits the closest with the numerical results of Kw SST. Comparing at a constant Reynolds number of 5000, the streamwise Nu distribution remains geometrically similar, but the Nusselt number decreases. It is this specific case which yields the highest disagreement, with a maximum average deviation of 32% between the RNG model and the Bunker correlation. Better agreement is obtained at the high Reynolds number of 15000, Figure 4.26:

Figure 4.26 – Streamwise Resolved Nu for $H/D=3$ at $Re_{Avg}=15000$

The agreement across the array with the Kw SST model is excellent, in contrast to the poor performance obtained with the RNG model.

The general trend throughout the different Reynolds numbers and jet to target ratios is that of a decreasing streamwise resolved heat transfer across the array. Both the Kw SST and RNG models are capable of capturing this decrease across the array, which is due to the diminishing strength of the impinging jets with increasing crossflow. Better agreement is seen at higher Reynolds numbers at both H/D values, with the furthest deviation produced by $H/D=3$ at $Re_{Avg}=5000$. Nevertheless, it is seen that either both or one of the turbulence models is capable of replicating the distribution within the range of the proven correlations. Both turbulence models predict the same streamwise resolved NU trend for all shown cases, the difference is the higher NU values predicted by the Kw SST. It is the higher prediction of heat transfer that situates the Kw SST in better agreement with the correlations. Overall, a good concurrence is obtained for the streamwise resolved Nusselt number, with the Kw SST turbulence model yielding a more accurate prediction.

4.5.4 Average Heat Transfer

The streamwise resolved Nusselt number is a good indication of the turbulence model being capable of capturing the increasing/decreasing trend across the array. In most cases, of outmost

concern is the overall average heat transfer which is produced by the array. Figures 4.27 and 4.28 illustrate the overall average Nusselt number for the numerical and cited experimental results.

Figure 4.27 – Average Nusselt number for $H/D=1.58$ at various Re_{Avg}

Figure 4.28 – Average Nusselt number for $H/D=3$ at various Re_{Avg}

The overall agreement is excellent. For both jet to target ratios the turbulence models are capable of capturing the increase in heat transfer with increasing Reynolds number. To better ascertain which turbulence model yields the overall better agreement, Table 4.5, summarizes the percentage

deviation of each turbulence model for all cases studied (a positive deviation indicates an over prediction):

Table 4.5 – Average Nusselt number deviation (%) for all validation cases

		$Re_{Avg}=5000$			$Re_{Avg}=15000$			Avg Dev
		Florschuetz	Bunker	Katti	Florschuetz	Bunker	Katti	
H/D =1.58	RNG	6.22	19.88	-----	3.37	9.19	-----	9.66
	K ω SST	0.36	14.71	-----	-7.01	-0.56	-----	5.66
H/D=3	RNG	26.56	32.27	17.64	17.84	18.98	17.97	21.88
	K ω SST	17.35	23.54	7.48	3.25	4.65	3.56	9.97

Clearly seen, the K ω SST model outperforms the K ϵ RNG model for all cases with the exception of the H/D=1.58 at $Re_{Avg}=15000$ when comparing to the Florschuetz correlation. Combining average deviation across H/D values, the overall average deviation for the RNG model is 15.7%, and 7.8% for the K ω SST model. Considering the three dimensionality and complexity of this type of flow regime, the deviation values are well within acceptable standards for numerical work in impinging systems. Furthermore, the objective of this study is not to accurately predict a specific value for design purposes, but rather to compare across numerical models to capture the overall enhancement. It can be concluded that the K ω SST model yields better agreement for modeling impinging arrays, and thus, it should be used for the full vane geometry and micro-jet assessment.

4.6 Conclusion

Two simplified validation vanes were modeled, meshed, and iterated using two well known turbulence models: K ϵ RNG and K ω SST. The strategic positioning of the impingement array in the flat section of the airfoil allowed an overall simplification by which the numerical results could be validated against experimental work. This was necessary as no experimental work has been performed on full vane geometry.

The two validation models had the same impingement array parameters: a jet to jet spacing of 4D, spanwise and streamwise, and an L/D ratio of 1.15. They differed in jet to target spacing, the base model having 1.58D, while a larger model having 3D. Different H/D ratios were required to assess the capability of the turbulence models to capture the flow and heat transfer differences created with varying parameters. For the same reason, each model was run at two Reynolds numbers characteristic of turbine cooling: 5000 for the low end, and 15000 for intermediary.

The crossflow to jet mass velocity ratio was found to be independent of Reynolds number - as expected from cited experimental work. A comparison to the Florschuetz cross flow distribution revealed excellent agreement for both turbulence models and at both jet to target ratios. The results indicate that both turbulence models are highly capable of capturing the hydrodynamics of the array, and should present no problem when full vane geometry is considered.

The local Reynolds number distribution was obtained for both validation models. The numerical results were capable of replicating the increasing velocity across the array. At the higher jet to target spacing, the predicted distribution was more uniform than at the lower one. This is in accordance with experimental trends. Stagnation point heat transfer was quite sensitive and could not be compared directly to experimental data. Instead, the average stagnation point drop with an increase in jet to target spacing was compared to Van Treuren's [31] study. An overall good agreement could not be obtained, where the minimum deviation achieved was 14.5%. Attributing to the sources of deviation for the very sensitive stagnation point heat transfer are: Plenum geometry & flow direction (Figure 4.4), and nozzle diameter. The overall trend for the stagnation point with changing geometry was captured, albeit, with a high deviation. On that note, the predicted stagnation point heat transfer should only be confided on when comparing across numerical models.

The streamwise resolved Nusselt number yielded excellent agreement with the experimental correlations. Both turbulence models predicted the same decreasing Nusselt number trend across the array for all cases iterated. Overall, due to the higher heat transfer predicted by the Kw SST model, it is concluded that it is the better predictive turbulence model for this type of impinging array.

Average heat transfer under the impinging array was as well studied for changing Reynolds number and jet to target spacing. All numerical results were capable of replicating the higher heat transfer obtained with a higher Reynolds number, and conversely, a lower heat transfer with an increase in jet to target spacing. In its entirety, validating against all correlations, the RNG model obtained an average deviation of 15.7%, while the Kw SST yielded only 7.8%. For better accuracy, it is therefore recommended to use the Kw SST model for the full vane geometry analysis.

When considering a full vane with impinging jets all around its inner surface, it is untrue to consider the flow more complex. The array modeled in this validation is considered "dense" -due to its relatively small spanwise and streamwise spacing, 4D. At such short streamwise spacing, the interaction

between the jets in the streamwise direction is significant. Dynamically, this leads to higher mixing, recirculation, turbulence, and thus heat transfer, which makes the numerical prediction difficult. The streamwise spacing in an actual vane, and the one which is modeled in this study (4.3), is much higher ($>7D$, increasing linearly across the array). When the separation is at that level, the interaction between the jets decreases and the flow dynamics is less complex. Therefore, the validation model studied in this section is considered suitable to establish confidence on the results obtained for full vane geometry.

5.0 HPT Vane – Macro Jets

The validation vane modeled in the previous chapter provided a basic understanding of flow behavior for an in-line impinging array. It was a necessary step, as was explained in the introduction, due to the lack of experimental work for realistic jet locations. In this chapter, the same validation geometry is utilized but modified with realistic jet locations. In doing so, the actual flow behavior and heat transfer inside a realistic vane are obtained.

5.1 Geometry & Modeling

The 2nd stage HPT vane, Figure 4.1, utilizes preferential cooling and an uneven distribution of impinging jets around its inner surface. In this analysis, the entire inner surface of the airfoil is given a constant temperature boundary condition, thus, rendering the strategic location of the impinging jets redundant. It is very common to find impinging arrays within turbine blades and vanes, which have linearly stretched jets in the streamwise direction: closely packed in the leading edge, with increasing streamwise separation distance towards the trailing edge. To avoid arbitrary jet locations, the same numbers of jets as the rene150 2nd stage HPT vane (Figure 4.1) are used; the initial starting locations of the jets in the suction and pressure surface are kept, but, a linear stretched pattern is used to position the jets towards the trailing edge. Figure 5.1 illustrates the jet locations around the plenum of the vane:

Figure 5.1 – Base vane plenum geometry and streamwise jet positions

6 rows of jets are located in the suction surface, while 7 are used in the pressure surface (from original HTP vane model, Figure 4.1). A linear stretch of $2.3D$ is used for the suction surface jets, and a lower value of $1.3D$ is used on the pressure surface jets. A total number of 42 jets are protruded on the plenum of the vane, which include three leading edge jets. Like the validation model, the jet to jet spanwise spacing remains constant across the vane at 4 diameters. The impingement surface, and outer geometry remained exactly the same as the validation model, see Figure 4.3. Only the positions and the number of jets on the plenum were modified. Figure 5.2 illustrates the plenum and spanwise spacing:

Figure 5.2 – Base vane plenum geometry and spanwise jet locations

The results from the validation model indicated that the trailing edge exit, rectangular cross section (see Figure 4.6), did not impact the heat transfer or any of the required results. To save on computational time, and to keep the heated airfoil surface area constant, the trailing edge exit channel was removed. Figure 5.3 illustrates the base vane model ($H/D=1.58$) before meshing:

Figure 5.3 – Base vane before meshing, $H/D=1.58$

As it can be seen, the trailing edge slot was simply cut. The behavior of the flow in this region of the airfoil was of no concern to this study, nor did it impact the heat transfer on the target surface.

To obtain the effect of increasing the jet to target spacing, an $H/D=3$ model was modeled as well. The same procedure as the validation model was used: extend the outer surfaces to obtain a higher jet to target spacing. A comparison of cross-section is shown below in Figure 5.4:

Figure 5.4 – cross-section comparison for base HPT vane at $H/D=1.58$ and $H/D=3$

The meshing procedure was identical to the validation model, thus the reader is referred to chapter 4, section 4.3.1 for details on the mesh creation. A depiction is included below for completeness, but for detailed locations illustrating boundary layers, and the impinging jets, the reader is referred to appendix A5.

Figure 5.5 – HPT base vane ($H/D=1.58$) complete mesh

5.2 Boundary Conditions and Numerical Setup

Figure 5.6 – HPT base vane ($H/D=1.58$) boundary conditions

Unlike the validation model, the analysis for the full vane geometry, with the larger diameter jets and smaller micro-diameter jets, are compared at a constant mass flow per unit of heated area, G . In order to keep the mass flow constant G across all models, the mass flow had to be specified at the inlet, rather than the pressure. Thus, the inlet face was set as a velocity inlet. The flow being incompressible; a constant inlet velocity yields the inlet mass flow required for a specific G ratio, thus, avoiding continuous iteration for inlet mass flow for given inlet pressure. To comply with the inlet boundary condition, the exit was changed to “outlet”, where the exit conditions are unknown and the solver iterates for them. No other differences exist between the set conditions for the full vane and the validation study, thus, the reader is referred to Table 4.2 for the specific boundary conditions and fluid properties.

5.3 Mesh Sensitivity

An extensive mesh refinement study was performed on the validation vane. The analysis set the required mesh resolution for the full vane, and thus a good indication of mesh element size was a ready known. The validation vane has a much higher density array, and therefore it is expected that the mesh resolution is accurate enough to capture the flow and heat transfer behavior for the full vane.

Nevertheless, comparisons are shown below for pressure drop, streamwise resolved heat transfer, and the local heat transfer distribution on the pressure surface.

The pressure drop across the jets gives a good indication of the general velocity distribution¹⁸. All three grids obtain the same distribution:

Figure 5.7 – Pressure drop across array for three different grids

The streamwise resolved heat transfer yields an excellent agreement across all three grids, Figure 5.8:

Figure 5.8 – Streamwise resolved heat transfer coefficient for three different grids

¹⁸ In this section, the parameters plotted are not fully explained as only the mesh sensitivity is of importance. When the results are discussed for each case, each plotted parameter will be explained in detail in their corresponding section.

The local heat transfer distribution is compared on the pressure surface, Figure 5.9. Good agreement is obtained since all curves overlap. Largest discrepancies are seen in the stagnation region – as expected.

Figure 5.9 – Local heat transfer distribution on pressure surface for three different grids

Like the validation vane, the difference between the fine and medium grid is not substantial, and therefore, to save on computational requirements, the medium grid resolution is chosen for the full vane analysis¹⁹.

5.4 Results

5.4.1 Pressure and Reynolds Number Distribution

The pressure distribution across the array is of great interest as it affects the exit velocity of the jets, the crossflow to jet mass velocity ratio, and therefore, the heat transfer efficiency of each jet row. Initial results are analyzed for the default case of $G=4.5 \text{ Kg/m}^2\text{s}$, where G is the mass flow per heated unit area. Table 5.1 summarizes key parameters for the initial full vane model:

Table 5.1 – Key parameters for full HPT vane at $G=4.5$, $H/D=1.58$

$G \text{ (Kg/m}^2\text{s)}$	Mass flow (Kg/s)	Inlet Velocity (m/s)	Plenum Pressure P_o (Pa)
4.5	0.003269	1.2266	1443903

¹⁹ The same local agreement is obtained for the larger jet to target spacing, $H/D=3$, model.

In the above table, the plenum pressure is not specified as a boundary condition, it is the resultant of the simulation itself, but it is given for completeness. A depiction of the absolute pressure across the vane is shown below, where the 2D plane cuts through the middle jet column:

Figure 5.10 – HPT vane absolute pressure contours (Pa), $H/D=1.58$, $D=0.51\text{mm}$, $G=4.5\text{ Kg/m}^2\text{s}$

The absolute pressure is highest in the plenum chamber; it is followed by a subtle decrease towards the trailing edge exit, where the pressure is lowest. Like the validation model, the pressure decreases towards the trailing edge due to the increasing crossflow velocities. The pressure at each jet exit can be obtained and averaged spanwise for each row. It is then normalized against the plenum pressure P_o , given in Table 5.1, and the pressure distribution for the suction and pressure surfaces is obtained, Figure 5.11:

Figure 5.11 – HPT vane pressure distribution, $H/D=1.58$, $D=0.51\text{mm}$, $G=4.5\text{ Kg/m}^2\text{s}$

The distribution is plotted with the actual non-dimensional curve length (C/D) values for each jet row (locations are shown with the markers). This allows a direct comparison to be made, regardless of row location or number. Also evident through the use of curve length is the longer length of the suction surface relative to the pressure surface.

The overall pressure drop across the suction and pressure surface jets are relatively the same; with final P/P_0 values reaching 0.9858 and 0.9853 in the last row of jets in the suction and pressure surfaces, respectively. Convergence of both values is to be expected as the plenum ends and both airfoil surfaces approach towards the trailing edge. Evident from Figure 5.11 is the higher pressure throughout the suction surface jets, and thus, a steeper pressure drop gradient for the pressure surface jets. This could be explained by the extra row of jets on the pressure surface: the overlap of the two curves occurs near the initial three jets on the pressure surface and the initial two on the suction surface. The presence of an added row adds to the continuous decrease of pressure throughout the jets. Therefore, the lack of a 7th row of jets in the suction surface yields a higher pressure at a given curve length from the leading edge. This effect is short lived as the overall pressure drop converges for both channels (suction and pressure) to the same value towards the trailing edge. To complement the higher pressure seen in the suction surface is also its longer channel length. In order for the pressure distribution to be continuous, as it should, the static pressure exiting the upper and lower channels must equalize (see Figure 5.10). Since the suction surface is longer, the rate at which pressure decreases must be lower than its counterpart in the pressure surface, only then can both values can converge at the trailing edge.

The lower pressure at a given curve length on the pressure surface should yield higher jet exit velocities than the suction surface jets at a given curve length from the leading edge. Figure 5.12 illustrates the velocity contours across the vane (through the middle jet column):

Figure 5.12 – HPT vane velocity contours (m/s), $H/D=1.58$, $D=0.51\text{mm}$, $G=4.5\text{ Kg/m}^2\text{s}$

Higher exit velocities are seen in the last row of jets, both in the suction and pressure surface. The maximum velocity occurs at the exit where the airfoil surfaces converge and accelerate the flow. Jet Reynolds numbers are plotted for a given non-dimensional curve length, Figure 5.13:

Figure 5.13 – HPT vane Reynolds number distribution, $H/D=1.58$, $D=0.51\text{mm}$, $G=4.5\text{ Kg/m}^2\text{s}$

As expected from the pressure distribution in Figure 5.13, at a corresponding curve length the pressure surface jets exhibit higher exit velocities, this is due to the lower jet exit pressure. The opposite is the case if the Reynolds numbers are compared on row basis: row 1 in the suction surface has a higher Reynolds number than row 1 in the pressure surface.

A better depiction showing the three dimensionality of the flow is given below; where the velocity contours are shown across planes streamwise and spanwise:

Figure 5.14 – HPT vane velocity contours (m/s) streamwise and spanwise, $H/D=1.58$, $D=0.51\text{mm}$, $G=4.5\text{ Kg/m}^2\text{s}$

Subtle differences exist between spanwise jet velocities. Maximum deviation occurs at the last row on the suction surface where the left jet (closest to the inlet) has a higher Reynolds number by 2.3% relative to the right jet. The differences can be attributed to the asymmetric inlet conditions, where the flow inlet is not perpendicular to the jet discharge (unlike most experimental work, see 2.2).

5.4.2 Crossflow

To determine the crossflow to jet mass velocity ratio, G_c/G_j , the same approach was used as the validation vane. The crossflow at a specific streamwise jet row is the average of the crossflow half a pitch upstream and half a pitch downstream from that specific jet row. Since the pitch is not constant across the array, as it was in the validation vane, the crossflow planes are not located at equal distances from the jet at all rows. An illustration depicting the locations of the crossflow planes can be found in appendix A6. To remain consistent with the previous crossflow model, the G_c/G_j distribution is plotted at a given row, and not at a curve length, Figure 5.15:

Figure 5.15 – G_c/G_j ratio for full HPT vane and validation vane (Fig 4.16), $H/D=1.58$, $D=0.51\text{mm}$, $G=4.5\text{ Kg/m}^2\text{s}$

Relatively, at a given row, the pressure surface rows experience less of a crossflow than the suction surface rows. What this entails is that the jet rows on the suction surface are influenced the most by the spent air, and thus, their jet impinging efficiency will decrease. Both distributions are similar to the validation vane, where the crossflow ratio increases nearly linearly across the array. The crossflow ratio was shown to be independent of Reynolds number, and therefore, it is plotted in Figure 5.15 for direct comparison with the validation vane. Clearly seen is the convergence of all curves as the row number increases, this could be due to the channel flow behavior developing with increasing row

number. The work of Florschuetz [31] demonstrated that the G_c/G_j ratio is independent of streamwise spacing, which is the main difference between the full vane and the one used for validation. Therefore, having the same spanwise spacing, and jet to target spacing should entail similar G_c/G_j distributions. The validation vane revealed that the K-w SST model was highly capable of replicating the Florschuetz distribution model, or G_c/G_j ratio, hence, the difference seen here is due to the geometry of the full vane. This could explain why more resemblance is obtained with the pressure surface distribution than the suction surface. The pressure surface channel is relatively flat compared to the highly cambered channel created between the plenum and the suction surface of the airfoil. As was seen earlier, at a given C/D , the pressure surface jets exit at a higher velocity, but at a given row, the suction surface rows experience a higher crossflow and higher jet flux, yielding an overall higher G_c/G_j at given row number. If the airfoil was symmetric, the distributions would be the same, but due to the curvature of the leading edge, the pressure gradients are not equal, yielding higher crossflow and jet fluxes on the upper channel. This effect is clearly seen in figure 5.15 where the deviation between the curves is highest in the early streamwise rows; an indication of the highly curved leading edge, followed by diminishing deviation between the distributions as the curvature lessens towards the trailing edge. Overall, the cross flow distribution of the full vane is relatively similar to a flat array of analogous geometry, with the largest deviation seen in the region of maximum curvature.

A look at the pathlines better illustrates how the spent air moves around the airfoil and sweeps downstream jets. Suction surface jets, Figure 5.16:

Figure 5.16 – HPT vanes suction surface pathlines, $H/D=1.58$, $D=0.51\text{mm}$, $G=4.5\text{ kg/m}^2\text{s}$

Pressure surface jets, Figure 5.17:

Figure 5.17 – HPT vanes pressure surface pathlines, $H/D=1.58$, $D=0.51\text{mm}$, $G=4.5\text{ kg/m}^2\text{s}$

The above figures should give an indication of the complex flow created by the mixing, and recirculation of the jets and spent air.

5.4.3 Local Heat Transfer

The heat transfer coefficient contours are illustrated below:

Figure 5.18 – HPT vane Heat Transfer coefficient contours ($\text{W/m}^2\text{K}$), $H/D=1.58$, $D=0.51\text{mm}$, $G=4.5\text{ Kg/m}^2\text{s}$

As it can be seen the heat transfer pattern on the inner airfoil surface is representative of the impinging jet array. Clear and distinct circular points of high heat flux are seen directly under the jets, followed by a drop like tail due to the crossflow. Similar footprints as the validation vane are obtained, with circular footprints when the crossflow is low (leading edge region) to stretched drop like figures near the trailing edge of the airfoil. In between streamwise regions indicate the magnitude of the crossflow: dark blue regions near the leading edge depict regions of low heat transfer, and light blue regions near the trailing edge indicate higher heat transfer due to forced convection of the channeled spent air. Also noticeable is the spanwise shrinkage of the impinging heat transfer footprint due to the sweeping crossflow: leading edge jets are clearly intertwined in the spanwise direction, trailing edge jets have a clear spacing in between them, where the crossflow is accelerated and to a certain degree aids in increasing heat transfer but diminishes impinging efficiency.

To extract the local heat transfer values on the curved surfaces of the airfoil, a program was coded via VBA and excel. The local heat transfer is defined at a resolution of 1D, via a strip that cuts through the middle column of jets. Figure 5.19 depicts the 1D strip (orange) whereby the heat transfer values are averaged spanwise and extracted to produce the local distribution through the airfoil (only suction surface strip shown). The leading edge local heat transfer is treated separately, and the extraction of local values begins slightly downstream of the immediate vicinity of the leading edge region (leading edge heat transfer is evaluated through the streamwise resolved Nu_x). The reader is referred to appendix A9 for a detailed explanation of the extraction process.

Figure 5.19 – 1D strip shown for local heat transfer extraction on full HPT vane

The non dimensional heat transfer, Nusselt number, has to be defined with a characteristic length which remains constant across all simulations. Since the diameter of the jet is decreased to ascertain the use of micro-jets on the HPT vane, it cannot be used, as this would skew the results towards the larger diameter jets. Instead, the Nusselt number is defined as follows:

$$Nu_L = \frac{hL_c}{k} \quad (5)$$

Where L_c is the characteristic length, chosen as the chord length of the airfoil: 66mm. The local Nusselt distribution for the suction impingement surface is plotted in Figure 5.20:

Figure 5.20 – Local Nu distribution on suction surface for HPT vane, $H/D=1.58$, $D=0.51\text{mm}$, $G=4.5\text{ Kg/m}^2\text{s}$

Several trends can be noticed in the above heat transfer distribution. The sharp stagnation points belonging to the typical fountain shape curve can be clearly seen. The symmetric fountain shape is only seen for the early jets, mainly the 1st row, crossflow then skews the curves downstream. Obvious in rows 2, 3, and 4, the upstream half of the stagnation region is swept downstream and adds to the downstream half, diminishing the symmetric nature of the circular jet (an effect experimentally observed in the work of Gao [38]). The dotted lines have been added at corresponding jet locations to better depict the effect of crossflow. It is interesting to note that although the crossflow changes the shape of the stagnation region, the stagnation point remains relatively constant directly under the jet. It is only at the 4th row that the stagnation point begins to shift downstream from its corresponding jet. In this case, the shift is quite small, only $0.8D$ downstream.

The regions in between the jets clearly illustrate the forced convection occurring, with the flat in-between regions progressively increasing in heat transfer across the array. Such was the case in Figure 5.18 with the lighter blue regions seen near the trailing edge. A last maximum is seen at the end of the suction surface, near 125 C/D, which is due to the accelerating exiting flow. The overall distribution is not uniform, that is, it does not decrease or increase from the initial jet. Maximum local heat transfer is observed in the 2nd row of jets, subsequently continuously decreasing through the array. The overall local decrease is attributed to the diminishing performance of the jets with increasing crossflow. The maximum heat transfer seen in the 2nd row is a product of not only the array geometry but jet diameter, Reynolds number, and exit flow direction. The sensitivity of the local Nu distribution is significant, and no functional correlation has been established for the variables which affect maximum and minimum local heat transfer.

The local Nu distribution on pressure surface is shown below, Figure 5.21:

Figure 5.21 – Local Nu distribution on pressure surface for HPT vane, $H/D=1.58$, $D=0.51\text{mm}$, $G=4.5\text{ Kg/m}^2\text{s}$

The Nu distribution for the pressure surface is relatively uniform for the initial 4 rows, followed by a drop and decreasing heat transfer from then on. The relatively closer streamwise spacing in the pressure surface could attribute to the more uniform distribution seen in the initial 4 rows. Compared to the suction surface jets, the pressure surface jets experience less crossflow at a given row, which could also aid in the steady stagnation regions in the initial four rows. As expected, due to the extra row of jets, the pressure impingement surface experiences an overall higher local heat transfer. Also noticeable is the decrease of heat transfer after the 7th jet, which does not occur for the suction surface (Fig 5.20).

Subtle differences can be attributed the different curvatures for each surface: Concave for the suction surface and convex for the pressure surface²⁰. No major boundary layer separation is seen at these speeds or curvature. Of course, the boundary layer behavior for crossflow-impingement cannot be explained by simple curvature, where it is known that a convex surface stabilizes the BL, and a concave surface destabilizes it (Logan and Ramendra [60]).

5.4.4 Effect of Mass Flow Increase

Increasing the mass flow per unit of heated area, G , will have the same effect as increasing the overall Reynolds number. Of course, it is more practical to see the change with increasing G , as mass flow directly affects the performance of the gas turbine. For the base vane of $D=0.51\text{mm}$ and $H/D=1.58$ three different G values were computed and analyzed: 2.25, 4.5, and 6. The reader is referred to Table 5.1 for the parameters of the $G=4.5$ case, Table 5.2 lists the parameters for cases with $G=2.25$ and $G=6$.

Table 5.2 – Key parameters for HPT Vane $H/D=1.58$ at $G=2.25$ and $G=6$

G ($\text{Kg}/\text{m}^2\text{s}$)	Mass flow (Kg/s)	Inlet Velocity (m/s)	Plenum Pressure P_o (Pa)
2.25	0.00163335	0.613303595	1437338
6	0.004359901	1.635476252	1450519

The increase of G has no effect on the crossflow distribution. Such was the case with the validation vane (no effect with increasing Reynolds number) and experimental work. Increasing the mass flow correspondingly increases the Reynolds number at each jet location. Figure 5.22 illustrates the effect of increasing G on the Reynolds distribution across the array:

²⁰ Considering the interior surface of the airfoil where the inner boundary layer forms, and not the external airfoil surface which would then be of opposite curvature

Figure 5.22 – Reynolds number distribution for HPT Vane $H/D=1.58$, $D=0.51\text{mm}$, at $G=2.25, 4.5, 6$

In the figure above, a negative axis was used to depict the rows on the pressure surface, making the origin the leading edge jet on the airfoil. An increase in mass flow linearly increases the Reynolds number at each jet location, keeping the shape of the base curve $G=2.25$. It is interesting to note that the leading edge does not experience the lowest Reynolds Number. Rather, a slight decrease in Reynolds number occurs on the initial row neighboring the leading edge jet, it then monotonically increases through the array. This phenomenon could be due to the unique position of the leading edge, whereby the impinging air can flow both ways, towards the pressure and suction surfaces. This was not the case in the validation vane, nor experimental work, where the spent air of the initial row can only flow towards one direction. As such, the initial row beside the leading edge assimilates the initial row in the validation vane, whereby its spent air is forced to exit only in one direction due to the spent air of the leading edge jet. It can then be said, that the leading edge jet yields a higher exit velocity than its neighbor due to the un-constrained exit conditions of its spent air.

The overall average Reynolds number for all pressure and suction surface jets increases linearly with increasing G . The Figure below illustrates the effect:

Figure 5.23 – Average Reynolds number with increasing G, HPT Vane H/D=1.58, D=0.51mm

Doubling the mass flow per unit of heated area (in this case, simply doubling the mass flow) correspondingly doubles the average Reynolds number. It is possible that the linear relationship between G and Re_{Avg} is only experienced in this G range. At higher G values, flow could choke through the jets and possibly channels, generating a more asymptotic distribution (peaking at a certain RE with a continuous increase in G). An increase in Reynolds number with increasing mass flow will consequently correspond to an increase in heat transfer. The local Nu_L distribution is plotted for the pressure surface for all three G values:

Figure 5.24 – Local Nu_L distribution on pressure surface, HPT Vane H/D=1.58, D=0.51mm, at G=2.25, 4.5, 6

As it can be seen the predicted distributions with increasing mass flow are simply shifted upwards. No anomalies (shape differences at higher G) are seen with increasing velocities, which

resembles the behavior observed with a single jet (see Figure 3.11). The same behavior is seen on the suction surface:

Figure 5.25 – Local Nu_L distribution on suction surface, HPT Vane $H/D=1.58$, $D=0.51\text{mm}$, at $G=2.25, 4.5, 6$

The similar curves obtained when increasing the mass flow would indicate a linear increase for the average heat transfer as well. The overall average heat transfer increases by 40% when doubling the G from 2.25 to 4. Figure 5.26 illustrates the average Nu_L increase with mass flow per unit of heated area, G :

Figure 5.26 – Average Nu on HPT Vane $H/D=1.58$, $D=0.51\text{mm}$ at various G values

The behavior is quite similar to the Reynolds number increase with increasing mass flow. A look at the heat transfer contours on the pressure side on the airfoil is shown below, where the color map was fixed for all G values so as to pronounce the heat transfer enhancement:

Figure 5.27 – Heat transfer coefficient contours ($\text{W/m}^2\text{K}$) for HPT vane, $H/D=1.58$, $D=0.51\text{mm}$, various G values

From the plots illustrating the local Nu distribution, and Figure 5.27, it can be seen that the heat transfer is enhanced mainly in the stagnation regions under the jets. Similar to the validation model, the trailing edge experiences an increase in heat transfer in the in-between streamwise regions of the jets, clearly evident as the mass flow increases (light blue regions on the trailing edge). Overall, the effect of increasing the mass flow is similar to increasing the Reynolds number for a single jet: it shifts the Nusselt distribution upwards, and increases the overall heat transfer on the target plate.

5.4.5 Effect of Increasing the Jet to Target Spacing

In the validation model, increasing the jet to target spacing to 3D reduced the overall crossflow to jet mass velocity ratio, G_c/G_j . The same behavior is seen for both surfaces in the full vane:

Figure 5.28 –Effect of H/D ratio on G_c/G_j ratio for HPT vane at $G=6 \text{ kg/m}^2\text{s}$

The decrease of the crossflow ratio with increasing jet to target spacing is significant. Relative to the flat validation array, increasing the jet to target spacing to 3D, reduced the overall average crossflow ratio by 80%, in this case, for the full vane, the reduction is similar at 76%. The distribution for both H/D values is more uniform on the pressure surface than the suction surface. This is evident with the semi-inflation point near the initial streamwise row (1), which could be due to the leading edge suction curvature. The distribution in Figure 4.56 is shown for the $G=6 \text{ kg/m}^2\text{s}$, but as was found earlier, the G or Reynolds number has no effect on the crossflow distribution.

The average Reynolds number increased linearly with increasing mass flow (Fig 5.25) for a jet to target spacing of 1.58D. Contrasting to $H/D=3$, the same pattern is obtained, but a higher increase is seen with increasing mass flow. Figure 5.29 plots the average Reynolds number for both jet to target ratios with increasing G :

Figure 5.29 – Average Reynolds number increase with G for HPT vane at H/D =1.58 and 3

Although the difference is small, at the same mass flow per unit of heated area, a higher average Reynolds number is obtained at a higher jet to target spacing. This trend is also observed in the experimental work of Florschuetz et al [31]. An increase in jet to target spacing would entail driving the same amount of flow through the orifice at a smaller diameter, since the mass flow is constant, the velocity through the orifice increases, yielding higher Reynolds number at a higher jet to target spacing. This of course is the explanation for a single jet, where crossflow does not interplay in the flow physics.

As was shown previously, in an impinging array, the crossflow undertakes a significant role in increasing exit velocities. Experimental work has shown that an impinging array at a higher jet to target spacing yields a more uniform velocity distribution [28-35]; the validation vane was capable of replicating this effect. A comparison of the Reynolds number distribution for both jet to target ratios at $G=6 \text{ kg/m}^2\text{s}$ is shown below:

Figure 5.30 – Reynolds number distribution for HPT vane at $H/D=3$ and $H/D=1.58$, $G=6\text{kg/m}^2\text{s}$

As it can be seen, an overlap of the curves occurs at the 4th streamwise row in both pressure and suction surfaces. As expected, the larger jet to target spacing $3D$, yields a more uniform velocity distribution across the array. Initially, a higher Reynolds number is obtained at the higher H/D value of 3, which is in accordance with what was previously explained. As the crossflow increases through the array, the exit velocities increase correspondingly. Having a higher crossflow to jet mass velocity ratio, the lower jet to target spacing overlaps the higher H/D value which experiences a lower crossflow (Figure 5.28). Thereby producing higher exit velocities downstream of the initial rows. The same trend is observed in the experimental work of Florschuetz et al [31], where different arrays overlap their velocity distributions near the 6th row.

The local heat transfer distribution on the pressure surface is compared for $G=6\text{ kg/m}^2\text{s}$ and for the suction surface at $G=2.5\text{ kg/m}^2\text{s}$. Figures 5.31 and 5.32, illustrate the change in local heat transfer with increasing jet to target spacing, for the pressure and suction surfaces, respectively.

Figure 5.31 – Local heat transfer distribution on pressure surface at $H/D=3$ and $H/D = 1.58$, $G=6 \text{ kg/m}^2\text{s}$

Figure 5.32 – Local heat transfer distribution on suction surface at $H/D=3$ and $H/D = 1.58$, $G=2.25 \text{ kg/m}^2\text{s}$

On the pressure surface, Figure 5.31, the local heat transfer diminishes with increasing jet to target spacing. The difference is subtle in the initial rows, where the stagnation region experiences the main decrease in heat transfer. The decreasing heat transfer with increasing jet to target spacing is also seen in the experimental work of Gao [38, 39], although average spanwise Nu is analyzed and not the local values. As the crossflow increases, in-between streamwise regions increase in heat transfer with the lower jet to target spacing having higher values. This is to be expected as $H/D=1.58$ has a lower channel height and higher crossflow velocities, adding to the forced convection seen in the exit rows. Such an effect is clearly evident in the last jet, where $H/D=1.58$ has a significantly higher Nusselt

number. Also noted is the exit heat transfer peak at the end of both distributions. With a lower jet to target spacing, the converging exit channel is much smaller, forcing the flow to accelerate at much higher speeds, producing higher heat transfer than at a higher jet to target spacing.

The suction surface experiences a similar change with increasing jet to target spacing. Two vertical lines are required for the jet positions since subtle changes in curve length are created due to the taller channel height and leading edge curvature (dashed vertical lines for $H/D=3$, and solid for $H/D=1.58$)²¹. For both surfaces, an increase in jet to target spacing does not have a major effect on the position of the stagnation regions. This is to be expected, as the larger channel height introduced by the increase in jet to target spacing, is counterbalanced by the lower crossflow experienced at these H/D values. Therefore, no significant effect is seen in terms of jet deflection for this particular H/D increase.

The overall average heat transfer decreases with increasing jet to target spacing. The same effect was found using the validation vane, and in the experimental correlations used as validation. Certain studies have found an optimal jet to target spacing near 1-3 H/D . The work of Brevet et al [35], obtained an optimal value of $H/D=2$, where anything larger would yield a decrease in heat transfer. In this study only $H/D=1.58$ and $H/D=3$ geometries were modeled, and thus, an accurate optimal value is not obtained, but the overall trend of decreasing heat transfer with increasing H/D is captured. Figure 5.33 compares the overall average heat transfer with increase G at both H/D ratios:

Figure 5.33 – Average heat Transfer for all H/D ratios with increasing G

²¹ There is no need for this on the pressure surface due to the significantly lower curvature. Therefore, no major differences are seen in terms of C/D .

An average 49% drop in heat transfer is experienced at all G values when the jet to target spacing is increased to $3D$. The increase with mass flow remains similar for both jet to target ratios, that is, approximately 39% when doubling the mass flow. The overall decrease can be attributed to the lower stagnation values and lower in-between streamwise regions experienced at the higher jet to target spacing - as was shown in the local distributions. What these results entail is that the vane designer should strive to keep the jet to target length as small as possible, of course, given the manufacturing constraints and stress considerations.

A look at the heat transfer coefficient contours on the vane at $G=2.25$ for both H/D ratios is shown below. No significant difference is seen between them, except for the higher non-uniformity seen at $H/D=3$ in the stagnation region, accompanying the decrease in heat transfer.

Figure 5.34 – Heat transfer coefficient contours (W/m^2K) for HPT vane at $G=2.25$, $H/D=1.58$ & 3

5.5 Conclusion

A full HPT vane was modeled, meshed, and solved using Fluent. A base case was analyzed at a mass flow per unit of heated area, G , of $4.5 \text{ kg/m}^2\text{s}$. The static pressure decreases across the vane, starting from the leading edge and monotonically decreasing towards the trailing edge exit. At a given curve length from the leading edge jets, the pressure surface jets experience a steeper pressure drop.

This effect has been attributed to the extra row of jets on the pressure surface, and likewise, the longer length of the suction surface yielding a lower pressure gradient. If the pressure drop across the jets is compared on a row by row basis, the surface jets experience a higher pressure, and thus, corresponding exit velocities at a given row number. The jet velocities increase uniformly across the array in both surfaces. The leading edge jet does not yield the lowest velocity as its spent air is free to move towards the pressure and suction surface channels. Instead, it was observed that the initial jet, next to the leading edge jet, is analogous to the initial jet on a confined jet array forced to direct its spent air in one direction (such as the one used for validation purposes, and most of the experimental work).

At a given row, the crossflow is highest in the suction surface channel. Due to the lower number of rows on the suction surface, higher mass fluxes for both crossflow and jets are seen at the corresponding rows, thus yielding a higher G_c/G_j ratio at each row – relative to the pressure surface rows.

The typical fountain shape Nusselt number distributions were obtained for the initial rows under the jets. As the crossflow increases across the array, the fountain shape decreased in uniformity and symmetry. The in-between streamwise jet regions were observed to increase in heat transfer progressively across the array—an evident effect of the forced convection in the impingement channel.

An increase in mass flow increased all local Reynolds numbers, yielding a linear curve for the overall array Reynolds number with increasing G . As was obtained in the validation vane and observed in experimental work, increasing the mass flow has no effect on the crossflow G_c/G_j distribution. The higher velocities obtained at a higher mass flow increased the local heat transfer distributions by simply shifting the curves upward. Doubling the mass flow correspondingly doubles the average Reynolds number across the array, and increases the average heat transfer by 40%.

At a constant mass flow, an increase in jet to target length from $1.58D$ to $3D$ consequently raises the average Reynolds number across the array. Locally, a more uniform Reynolds number distribution is obtained at a higher jet to target spacing. As such, the Reynolds distribution for $H/D=1.58$ crosses the $H/D=3$ distribution near the 4th streamwise row. This effect has been attributed to the higher crossflow experienced at a lower channel height, thus yielding higher the exit velocities at lower H/D ratios downstream of the initial rows.

The local heat transfer distribution is shifted downwards with an increase in jet to target spacing. More specifically, higher heat transfer is obtained in the stagnation and in-between jet regions at a lower jet to target spacing. Overall, an average 49% drop is experienced in average heat transfer when the jet to target spacing is increased to 3 diameters. For that reason, it is desirable to have a low jet to target spacing in an HPT vane such as this one.

6.0 HPT Vane – Micro Jets

The key objective of this study is to analyze the enhancement attained with the use of micro-jets in a HPT vane. The previous chapter provided a detailed analysis of the flow behavior inside a realistic HPT vane. Chapter 6 now models the same vane but with micro-diameter jets of 0.25mm. Comparisons of the velocities, crossflow ratios, local and average heat transfer coefficients, will reveal the differences obtained when micro-jets are used to cool the vane.

6.1 Geometry and Modeling

To determine the enhancement of using micro-jets in the base vane analyzed in the previous chapter, two different models using a jet diameter of $D=0.25\text{mm}$ are analyzed. The first model, A, is a simple reduction in jet diameter from the default vane. As such, it is the resultant model if the base vane was to be drilled with orifices of $D=0.25\text{mm}$ instead of $D=0.51\text{mm}$. At the same time, the same jet to target spacing of $H/D=1.58$ is kept, which would entail welding the baffle to the support ribs at a closer length. This is done to keep a consistent comparison at the same jet to target ratio, which is a dominant variable in impinging arrays. A consequential side effect of decreasing the diameter of the jets is the increase in non-dimensional pitch in all directions. In the streamwise direction, the effect would be minimal as the streamwise pitch for the base vane is already quite high, and thus, the behavior resembles that of a row of jets in crossflow, rather than a dense array such as the validation vane. Also, the vane has to remain of the same scale and geometry across the different models, this would allow for direct comparisons for the use of micro-jets. Forcing the same streamwise spacing across all models with changing diameter would require increasing the number of jets, or changing the positions of the jets themselves around the vane, in both cases, those changes are outside the scope of this study. Also note, that the dimensional spacing remains the same, it is simply the non-dimensional spacing X_n/D , Y_n/d that changes across the models, since the diameter is shrunk. Although the non-dimensional streamwise spacing is allowed to change, a 2nd model, B, is also created to replicate the same non-dimensional spanwise spacing as the base vane, $4D$. In this model, the same base vane is used, but the jets are shrunk and brought closer together in the spanwise direction, attaining a spanwise spacing of $4D$. The main purpose of this model is to ascertain if there are any differences with changing spanwise spacing, which would add to the enhancement seen via micro-jets only.

Model A at $H/D=1.58$ is depicted in Figure 6.1. As it can be seen and as was explained, the jet locations are exactly the same as the base vane, only differences are the smaller jet diameters and the channel height which is reduced to keep the same jet to target spacing:

Figure 6.1 – Model B depiction, $D=0.25\text{mm}$, $H/D=1.58$, spanwise spacing = $8.16D$, $L/D = 2.142$

Model C is depicted below, where it can be seen how the jets are brought closer together to replicate the non-dimensional spanwise spacing of $4D$:

Figure 6.2 – Model C depiction, $D=0.25\text{mm}$, $H/D=1.58$, spanwise spacing = $4D$, $L/D = 2.142$

A summary of the geometrical parameters for each model is given in Table 6.1:

Table 6.1 – Geometrical parameters of HPT vane models

	D (mm)	H/D	Spanwise Pitch (D)	Streamwise pitch (D)	L/D
Base Model A	0.51	1.58	4D	Base	1.05
	0.51	3	4D	Base	1.05
Model B	0.25	1.58	8.16D	Base x2	2.142
	0.25	3	8.16D	base x2	2.142
Model C	0.25	1.58	4D	Base x2	2.142
	0.25	3	4D	Base x2	2.142

Noticeable from the table above is the change of orifice length to diameter ratio (L/D). The base vane was created from the Halila et al [8] vane model, yielding an L/D of 1.05. As the diameter is reduced by half (~ 0.49), the ratio L/D increases, thereby elongating the length of the jet itself. The experimental work of Garimella and Nenaydykh [41] studied the effect of L/D on the heat transfer under a single semi-confined jet. The results indicated that the heat transfer decreases as the L/D increases, but in the range of $1 \leq L/d \leq 4$ the heat transfer is not significantly affected. Also, the major deviation with changing L/D occurs in the stagnation region with the average heat transfer not greatly changed. Therefore, it is expected that the L/D increase for the micro-jets will have no noticeable effect in the full vane analysis, and if there is, it would be a minor effect which would actually decrease the overall heat transfer for models B and C²².

For completeness, key flow parameters for all three models are summarized in Table 6.2 and 6.3, for $H/D=1.58$, and $H/D=3$, respectively. Subtle differences are seen in mass flow requirements for the same G values. This is due to the different channel heights and curved geometry, producing very small differences in area towards the trailing edge²³. For that reason the analysis is done at a constant mass flow per unit of heated area, and not just the inlet mass flow. G is a common parameter when comparing numerous and realistic impinging arrays, an example of its use can be found in the experimental work of Andrews et al [40].

²² Fixing the L/D ratio across all models would have required changing the plenum-inlet face area, thereby producing a greater side-effect than a small increase in L/D .

²³ Offsetting the target impingement surface (inner vane surface) to a larger channel height changed the length of the curve on the trailing edge. To keep the outlet area consistent, a decrease in the trailing edge surface area was required.

Table 6.2 – Key flow parameters for all three HPT vane models of H/D=1.58

H/D = 1.58				
	G (kg/m²s)	Mass Flow (kg/s)	Inlet velocity (m/s)	Plenum Pressure Po (Pa)
Base Model A	2.25	0.0016	0.6133	1437338
	4.5	0.0033	1.2266	1443903.6
	6	0.0044	1.6355	1450519.2
Model B	2.25	0.0016	0.5937	1489083.3
	4.5	0.0032	1.1875	1642174.9
	6	0.0042	1.5833	1798334.2
Model C	2.25	0.0016	0.5939	1489126
	4.5	0.0032	1.1878	1642632.5
	6	0.0042	1.5837	1799301.6

Table 6.3 – Key flow parameters for all three HPT vane models of H/D=3

H/D = 3				
	G (kg/m²s)	Mass Flow (kg/s)	Inlet velocity (m/s)	Plenum Pressure Po (Pa)
Base Model A	2.25	0.0017	0.6278	1438164.3
	4.5	0.0033	1.2557	1447038.3
	6	0.0045	1.6742	1455977.2
Model B	2.25	0.0016	0.6120	1505479.5
	4.5	0.0033	1.2240	1705760.5
	6	0.0044	1.6321	1910910.2
Model C	2.25	0.0016	0.6120	1500964
	4.5	0.0033	1.2241	1688244.2
	6	0.0044	1.6321	1879704.3

The boundary conditions and numerical setup for all micro-jet models are left exactly the same as the base vane analysis in chapter 5, thus the reader is referred to 5.2 for the specific details. Small changes had to be implemented on the mesh so as to obtain an accurate resolution. Since the diameter of the jets is decreased, so did the grid resolution. The mesh sensitivity has been studied for the base vane in chapter 5, section 5.3; the reader is referred to appendix A7 for the local mesh sensitivity plots for models B and C.

6.2 Results

6.2.1 Reynolds Number Distribution and Crossflow

The exit velocity of the micro-jets must be higher than their larger diameter counterpart. At a constant mass flow per unit of heated area, the same amount of air is driven through the orifices, and therefore the velocity is expected to increase. The figure below illustrates this effect for a jet to target spacing of $H/D=1.58$ and $G= 4.5 \text{ kg/m}^2\text{s}$:

Figure 6.3 – Reynolds number distribution for HPT Vane models at $G=4.5 \text{ kg/m}^2\text{s}$, $H/D=1.58$

At a constant mass flux, halving the diameter of the jets to 0.25mm doubles the average Reynolds number across the array. The local streamwise distribution remains similar, but a relatively flatter curve is seen across the array. This is evident since, for the large diameter array, the Reynolds number increases by 29% from the initial suction row to the last one. For both models B and C with $D=0.25\text{mm}$ this increase is only 12%. The two small diameter vanes obtain similar velocity distributions although parametric differences exist between them. In this case, the spanwise spacing between the jets does not affect the streamwise resolved exit Reynolds number. At a higher jet to target spacing of $H/D=3$, the same effect of higher Reynolds number is obtained, Figure 6.4:

Figure 6.4 – Reynolds number distribution for HPT Vane models at $G=4.5 \text{ kg/m}^2\text{s}$, $H/D=3$

As was shown with the analysis of the large diameter vane in chapter 5, a flatter distribution is obtained when increasing the jet to target spacing. The same trend is obtained for the small diameter vanes at a higher jet to target spacing. The overall average Reynolds number also increases with a higher jet to target spacing, as was the case with the larger diameter vane analysis. Also noted is the increase in sensitivity for the small diameter vanes, this is evident at both H/D ratios, where the distributions for the $D=0.25\text{mm}$ jets are not fully uniform as their larger diameter counterpart. It is possible that as the diameter decreases, the free jets are more susceptible to the curvature of the vane, thus producing uneven velocity distributions which are not seen with a larger diameter jet. The overall increase of Reynolds number with the decrease in diameter is significant, and it is therefore expected to have a major impact on the heat transfer produced.

The crossflow to jet mass velocity ratio G_c/G_j is compared for all three models at both H/D ratios. Figures 6.5 and 6.6 illustrate the change across the different models.

Figure 6.5 – G_c/G_j ratio distribution for all HTP Vane models at $G=4.5 \text{ kg/m}^2\text{s}$, $H/D=1.58$

Figure 6.6 – G_c/G_j ratio distribution for all HPT Vane models at $G=4.5 \text{ kg/m}^2\text{s}$, $H/D=3$

Clearly evident is the significant decrease in crossflow for the smaller jet vanes. Florschuetz et al [31] experimental work has shown that the crossflow ratio is independent of the streamwise spacing. A key difference between the larger diameter vane and micro jet vanes is the streamwise spacing, which is doubled as the diameter is halved. Since this spacing does not affect the crossflow ratio, the decrease seen in the above figures cannot be attributed to the streamwise factor. What does play a key role is the

spanwise spacing, which Florschuetz et al [31] has shown to be a dominant parameter. A larger spanwise spacing decreases the crossflow ratio through the array due to the decrease crossflow acceleration in the spanwise in-between jet regions. Therefore, model B with $D=0.25$ and a spanwise spacing of $8.16D$ should have a lower crossflow ratio across the array than the base model. This of course is a consequential benefit of decreasing the diameter of the jets, as a lower crossflow increases the jet impinging efficiency. The interesting case is model C, which technically has the same spanwise spacing as the base model, $4D$. Yet, the crossflow ratio significantly decreases relative to the base model, and overall yields the same distribution as model B. The distribution model of Florschuetz (see 2.3), which the turbulence model was highly capable of replicating, is geometrically dependent on the jet to target distance and spanwise spacing. Given those two variables, model B should yield a similar crossflow ratio across the array as the base model, but these results clearly indicate that the overall channel spanwise width is a dominant variable on top of the spanwise spacing. In the experimental work of Florschuetz et al [31], and others (Van Treuren [28], Bailey and Bunker [33]) the number of spanwise jets is increased in order to achieve a specific streamwise spacing. With this method, the model remains geometrically similar in the spanwise direction, that is, the distance from the walls to the nearest jet is half a pitch, regardless of the spanwise spacing. In model C, the jets are brought together to replicate the spanwise spacing, but the number of jets remains constant, and so does the channel width. In doing so, the behavior of the flow in between the spanwise jets changes - In this case, the spent air of upstream rows is allowed to navigate around the jet row itself. For that reason, the crossflow ratio remains relatively unchanged to model B, and significantly lower than the base model - The flow is not forced to go through the jets, but it can move around it, decreasing the overall G_c/G_j distribution. The results indicate that the distribution model can only be used when the end walls are situated half a pitch from the near wall jets, if this is not the case, the channel width to spanwise spacing ratio could provide a better parameter to correlate the crossflow distribution.

Figure 6.7 illustrates the crossflow patterns on the suction surface for base model A and model C at $H/D=1.58$, $G=4.5 \text{ kg/m}^2\text{s}$:

Figure 6.7 – Pathlines illustrating crossflow pattern for base model A and model C, $H/D=1.58$, $G=4.5 \text{ kg/m}^2\text{s}$

As it can be seen above, the crossflow pattern is quite different when the jets are brought together and the spacing between the walls is increased. The relative increase in length in the near wall region allows the flow to behave as if the three jets are combined, nearly the behavior of a slot jet, thereby explaining the low crossflow ratio obtained in Figure 6.5.

A low crossflow ratio typically induces higher jet impingement efficiency. The analysis of the base vane in chapter 5 revealed that an increase in jet to target spacing decreased the overall crossflow experienced, and in doing so, the overall heat transfer was reduced. The reason behind this was due to the increase in jet to target spacing, which reduced forced convection, heat transfer in the stagnation region and increased jet deflection. In this case, a decrease in diameter for the micro-jets reduces the overall crossflow ratio while still increasing the jet velocities, and at the same time, the low jet to target spacing is kept. For those reasons, it is expected that the heat transfer should increase.

6.2.2 Local Heat Transfer

The local Nusselt distribution is plotted below for the suction and pressure surfaces at $G=4.5 \text{ kg/m}^2\text{s}$:

Figure 6.8 – Local Nusselt distribution on pressure surface for all three HPT Vane models, $H/D=1.58$, $G=4.5 \text{ kg/m}^2\text{s}$

Figure 6.9 – Local Nusselt distribution on suction surface for all three HPT Vane models, $H/D=1.58$, $G=4.5 \text{ kg/m}^2\text{s}$

The obvious enhancement of local heat transfer is seen in both surfaces. On the pressure distribution plot, a single vertical line indicates the positions of jets. For the suction surface plot, two vertical lines indicate the positions, with red indicating jet locations for model B & C, and blue indicating the locations the base model A. The need for two vertical lines originates from the curvature of the leading edge. Since the suction surface channel height for the small diameter vane is shorter than the

one for the base model, the curve length to a jet location is slightly longer for the base vane. Therefore, to avoid skewing the graph, two vertical lines were plotted for each corresponding jet location. The pressure surface channel is not as highly curved as the suction channel, and therefore, only one jet location was required. For the horizontal axis, the dimensional curve length was used since the non-dimensional form C/D relies on the diameter which varies across the models.

Aside from the significant heat transfer enhancement seen at the same mass flow for the micro-jet models B and C, other key characteristics are seen in the above graphs. One of them is the highly uniform distribution obtained when using micro-jets. In Figure 6.8, it is noted how the base model decreases in heat transfer with increasing streamwise distance. In the earlier analysis, this was attributed to the crossflow deflecting the downstream jets thereby reducing their penetration into oncoming flow. The benefits of the micro-jets is therefore multi-faceted: they reduce the crossflow by increasing the spanwise spacing, higher exit velocities increase the penetration of the jet, and thus lessen the effect crossflow induces, and consequentially, this yields a more uniform/steady heat transfer distribution on the airfoil surface. When decreasing the diameter of the jets and keeping the same jet to target ratio, the channel height is actually reduced, since the diameter of the jet is decreased. Therefore, the forced convection channel effects are not heavily impacted and are relatively enhanced due to the higher velocities obtained. This effect is particularly seen on the pressure surface (Figure 6.8) where the in-between streamwise jet regions experience higher heat transfer for models B and C.

Models B and C do not differ in the local Nusselt number distribution. For surfaces, pressure and suction, the curves overlap throughout the array. The results indicate that the local distribution experienced by the middle jet (local distribution is a 1D strip cutting through the middle column of jets) is independent of the spanwise distance of its neighboring jets. The middle jet for model A is situated $8.16D$ from the near wall jets, for model B the spanwise distance is halved to $4D$. At such low jet to target lengths, the effect the jet has on its neighbor is quite limited, and it is mainly seen streamwise in terms of its spent air. The single jet analysis revealed that a confined jet would produced toroidal vortices near $2.5D$ from the stagnation point at $H/D=1$ (see Figure 3.5 and 3.6), and its heat transfer footprint would typically not go beyond $5D$ from its center. As such, spanwise jet to jet interaction should not be significant unless the spacing is less than $4D$. Furthermore, any changes that do occur with streamwise spacing will not captured with a 1 diameter strip through the middle jet, as this basically represents the stagnation region. It is therefore expected that decreasing heat transfer with increasing spanwise spacing does occur, but the effect should not be visible locally near the stagnation region.

Figure 6.10 illustrates a comparison of the heat transfer coefficients for $H/D=1.58$ at a constant G value of $4.5 \text{ kg/m}^2\text{s}$:

Figure 6.10 – Heat transfer coefficient contours ($\text{W/m}^2\text{K}$) for all HPT vane models at $H/D=1.58$, $G=4.5 \text{ kg/m}^2\text{s}$

The clear distinction between the models can be seen in the near stagnation regions. The smaller footprints for the micro-jets are evident, and the single-row like behavior is noted for model C. The above figure basically illustrates that the major enhancement of heat transfer using micro-jets occurs due to the higher exit velocities, hence, in the near stagnation regions under the jets.

6.2.3 Streamwise Resolved Heat Transfer and Overall Average Heat Transfer

The streamwise resolved average heat transfer was used for comparison with the experimental work in the validation vane. It is defined as the average heat transfer under the jet: half a pitch in all directions, spanwise and streamwise, and then averaged across the span for all spanwise jets. Since the streamwise pitch is not constant for the full HPT vane, the analysis is not the same as was done for the validation vane. Nevertheless, the average streamwise resolved heat transfer gives a more complete

picture of the overall heat transfer across the vane, than the local values across the middle jet column. For the full vane, the streamwise resolved heat transfer is defined on a row basis, where the row is a 1 diameter strip *spanwise* at that row jet location. Row 1 thus indicates the average spanwise heat transfer for the initial row of jets. Row 1.5 indicates the average spanwise heat transfer for the patch between row 1 and 2. The reader is referred to appendix A8 for an illustration depicting the different regions for the average streamwise resolved heat transfer.

Figures 6.11 and 6.12 plot the streamwise resolved Nu_L for $H/D=1.58$, and $H/D=3$, respectively, at a constant G of $4.5 \text{ kg/m}^2\text{s}$.

Figure 6.11 – Streamwise resolved Nu_L for all HPT vanes at $H/D=1.58$, $G=4.5 \text{ kg/m}^2\text{s}$

Figure 6.12 – Streamwise resolved Nu_L for all HPT vanes at $H/D=3$, $G=4.5 \text{ kg/m}^2\text{s}$

In the above figures the origin represents the leading edge jet heat transfer, and with the same convention used before, the negative axis plots the pressure surface distribution. At both H/D ratios the base model experiences significant jet deflection after the 3rd row of jets, thereby reducing the average heat transfer. For H/D=1.58, the average heat transfer decreases 50% from the leading edge row to the 6th exit row on the suction surface, for model B this decrease is only 23%. This large decrease is evident, as the peaks can no longer be seen after the 3rd jet row, and further downstream, the jets are deflected more than 1D, thereby producing a minimum instead of a maximum at that jet row location (note the 5th suction jet row at both H/D ratios, a decrease is obtained instead of an increase in heat transfer). Although it has been noted that the main increase in heat transfer occurs on the stagnation region, it can be seen from the above two figures that the uniformity of the average distribution adds to an overall higher average heat transfer for the micro-jets.

Relative to model B, a slightly lower streamwise resolved heat transfer is experienced by model C with the closer jet to jet spanwise spacing. Closer spanwise spacing yields higher in-between spanwise regions of heat transfer, but the near wall regions are left untouched by the jets themselves (see Figure 6.10), which reduces the overall heat transfer in that region. The larger spacing between the jets for model B has a larger coverage of the target surface, and thus, the air spreads a larger area increasing the overall heat transfer. In this case, both effects equalize yielding similar distributions, with the visible deviation occurring in the stagnation region as model B yields higher stagnation heat transfer.

The average heat transfer is obtained for the entire airfoil surface: Table 6.4 summarizes the values for all H/D ratios and mass flow constants G.

Table 6.4 – Average Nusselt number for all HPT vane models at various H/D ratios and G values

G (kg/m ² s)	H/D	Nu _{Avg}		
		Model A	Model B	Model C
2.25	1.58	1995.224	5456.678	4969.741
	3	1676.912	4494.448	4258.459
4.5	1.58	3321.281	9066.653	8399.277
	3	2746.224	7739.412	7376.813
6	1.58	4165.984	11191.99	10330.26
	3	3417.087	9589.051	9125.273

For brevity and clarity, the values are plotted below, Figures 6.13:

Figure 6.13 – Average Nusselt number with changing G and H/D, all HPT vane models

A decrease in heat transfer occurs for all models and at all mass fluxes with an increase in jet to target spacing. Compared to the base model and regardless of the jet to target spacing, model B enhances the average heat transfer by 63%, and model C by 61%. It is interesting to note that even at the lowest mass flow per unit of heated area, $G=2.25 \text{ kg/m}^2\text{s}$, model C and model B yield a higher average heat transfer value than model A at the highest G value of $6 \text{ kg/m}^2\text{s}$. Therefore, if optimizing for a specific average heat transfer value on the inner surface of the vane, the micro-jet vane will yield a higher average heat transfer with less bleed flow. A closer look at these results entails an analysis of the pressure drop across the vane. The focus is set only on model A and model B since model C yields similar values to model B but lower. The overall pressure drop is analyzed in percentage, more specifically: the % drop of absolute pressure from the plenum (P_o) to the trailing edge (P_{Te}) exit:

$$\Delta P = \left(\frac{P_o - P_{Te}}{P_o} \right) \% \quad (6)$$

In many cases a designer finds it preferable to obtain the highest heat transfer coefficient, and thus cooling, giving a fixed pressure loss percentage (Andrews et al [40]), or impingement pressure ratio, P_o/P_{Te} , Halila et al [8]. Figure 4.74 illustrates that at a constant mass flux a significant enhancement occurs when using the smaller diameters jets, but the pressure drop percentage required to yield that mass flux through the smaller jets is also significantly higher. Figure 6.14 plots the pressure drop % for a given mass flow per unit of heated area, G :

Figure 6.14 – Pressure drop across HPT vanes with increasing mass flow, $H/D=1.58$

As it can be seen, an increase in mass flow increases the pressure drop % across the vane. This is to be expected as the jet velocities increase and thus increase the pressure losses associated with the higher velocities (viscous). In the overall sense, at a constant mass flow, an enhancement of 63% in average heat transfer is obtained with model B, but this requires an increase of 75% in pressure drop percentage: Model A at $G=4.55 \text{ kg/m}^2\text{s}$ yields a ΔP of 4.29%, Model B has a much higher ΔP of 17.6%. In this type of application, an HPT vane using only impingement cooling, the pressure drop is not of outmost importance. There is usually enough pressure to choke the flow through the impingement orifices, and thus the objective is to attain as high of a heat transfer as possible with a given bleed air mass flow (Zuckerman and Lior [36]). Yet, there may be instances where the pressure loss is quite important, such as a vane with film cooling (1st stage HPT vane, Figure 1.1), or a rotor blade with numerous rib roughened serpentine passages. In that instance, a positive backflow margin must be guaranteed in order to drive the flow through the film holes and protect the outer surface. Or, in the case of a rotor blade, the impingement baffle pressure for the leading edge jets might be low due to the head loss as the air channels through the serpentine ribs. For those reasons, the plot below illustrates the average Nusselt number obtained for models A and B, with increasing pressure drop %, Figure 6.15:

Figure 6.15 – Average Nusselt number for HPT vanes with increasing pressure drop, $H/D=1.58$

As it can be seen, the direct comparison at a constant mass flux is a bit unfair, as the values were significantly higher than the base model with respect to the pressure loss %. The analysis in this study was to evaluate at a constant mass flow, and thus only three values were analyzed. Yet, it can be seen above, that only the highest G value for model A attains the same pressure loss as the lowest G for model B. Therefore, it can be seen that at a constant pressure loss %, the vane using the micro-jets yields higher heat transfer than the base vane with the larger jets (i.e. at ~5% ΔP , model B yields a much higher average Nusselt number)

A power trend line was used to correlate Nu_{Avg} vs ΔP . This is in accordance with the behavior of the Reynolds number with increase in pressure drop²⁴. From Figure 6.15 the following two expressions can be extracted for the average Nusselt number, where A and B subscripts denote model A and B, respectively:

²⁴ In the validation vane analysis several plots were extracted to attain a specific mass flow for given inlet pressure. The curves followed a power trend line such as Figure 6.15. Likewise, it is to be expected that 0 ΔP yields no mass flow and therefore no heat transfer. It is also customary to use a power function to correlate the Nusselt number against Reynolds number (see 2.1 and 2.2), or in this case, pressure drop (since it is directly proportional to the Reynolds number).

$$\overline{Nu}_A = 1926\Delta P^{0.378} \quad (7)$$

$$\overline{Nu}_B = 2714\Delta P^{0.421} \quad (8)$$

If a 6% pressure drop is chosen for certitude(extrapolation outside of iterated values does not have to be done), and equations 7 and 8 are used to obtain the average Nusselt number, 3791, and 5770, are obtained for models A and B respectively. In this case, the micro-jet vane provides a **34.3%** heat transfer enhancement compared to the larger diameter vane. Hence, it is evident that the micro-jets enhance heat transfer at a constant mass flux and set pressure loss %.

If of interest is to decrease bleed mass flow so as to increase the overall efficiency of the turbine cycle, but attain a specific average heat transfer coefficient, a similar enhancement is obtained using the smaller diameter vane. From Figure 6.14, the following expressions can be extracted for the pressure drop % at a given mass flow:

$$\Delta P_A = 0.220G_A^{1.974} \quad (9)$$

$$\Delta P_B = 1.280G_B^{1.735} \quad (10)$$

Equations 9 and 10 can be substituted into 7, and 8 to yield the average Nusselt number as a function of G. G can then be solved to find:

$$G_A = \left[\frac{\overline{Nu}_A}{1926(0.220)^{0.378}} \right]^{1/0.7461} \quad (12)$$

$$G_B = \left[\frac{\overline{Nu}_B}{2714(1.280)^{0.421}} \right]^{1/0.7304} \quad (13)$$

Equations 12 and 13 yield the required mass flow per unit of heated area (Kg/m²s) for models A and B. If an average Nusselt number of 4000 is desired, $h_{Avg} = 2441 \text{ W/m}^2\text{K}$, the large diameter vane requires 5.734 kg/m²s, and the micro-jet vane only 1.473 kg/m²s. Therefore, at a constant average heat transfer, the use of micro-jets in this vane reduces the mass flow by **74.3%**. Equations 12 and 13 can be plotted with increasing average Nusselt number, and it is

clearly evident that the % reduction in mass flow between the models remains constant at approximately 74 %. Note that this is data at a constant pressure drop %, Figure 6.16.

Figure 6.16 – Mass flow per unit of heated area with increasing average Nusselt number, $H/D=1.58$

Overall, although the micro-jet array requires a higher pressure to achieve the same mass flow. The micro-diameter array does not need such a high mass flow, as it can produce the required heat transfer at a much lower mass flow rate. In another perspective, for a G value of 6 kg/m^2s , a 19% increase in plenum pressure is required for the micro-jet array, but, if this increase cannot be afforded, then the mass flow can be reduced for the micro-jet vane to 2.25 kg/m^2s and it still yields a 23% increase in the average heat transfer rate compared to the larger diameter vane.

When considering the results obtained, certain key factors must remain in thought when considering this analysis and overall micro-jets. The array geometry remained the same throughout the study, and therefore the number of holes per unit surface area as well. Consequently, the overall open area ratio, A_r , (jet exit area to heated area ratio) decreased significantly for the micro-diameter jet vane. This could induce plugging of the holes, if any dust or sand is ingested. Likewise, to cover the entire vane surface – full coverage – decreasing the diameter of the jets would entail increasing the number of jets; otherwise the spanwise spacing would increase up to a point where the jets would not provide enough heat transfer to the in-between regions. In this study it seems that point was not reached, and the increase in spanwise spacing did not reduce the overall heat transfer. Increasing the number of jets, while reducing the diameter needs to be further explored so as to maintain the same open area ratio, which dominates pressure loss % and sand/dust plugging. With an increase in the number of jets, the pressure drop is expected to decrease, and so is their exit velocity, which would then decrease the

average heat transfer. As such, further research should look into attaining a balance between the number of jets and the jet diameter for full coverage impingement cooling, without having them directly depend on each other.

6.3 Conclusion

To ascertain the heat transfer enhancement of using micro-jets in an impingement only HPT vane, two different geometries were modeled, meshed, and solved. The two models consisted of jet diameters of 0.25mm and similar geometrical parameters to the base vane analyzed in the previous chapter 4, section 4.4. The results indicated that halving the diameter of the jets to 0.25mm doubles the overall average Reynolds number across the array, and reduces the Reynolds number gradient across the array. A significant decrease in crossflow occurs when the diameter is decreased. The reduction is attributed to the increase in non-dimensional spanwise spacing which occurs when D is halved. In the case of model C, where the jets are brought closer together to maintain the original spanwise spacing, the crossflow ratio decreased significantly as well, yielding a very similar distribution to model B. Extraction of the spent air pathlines indicated that the upstream crossflow navigates around the jet row, this due to the increased spacing of the end walls, thereby reducing the effective crossflow experienced by the downstream jets.

Comparisons of the local heat transfer in the middle column of jets revealed a much more steady distribution for the micro-jet vanes. This effect was attributed to the reduced crossflow by the increased spanwise spacing, less jet deflection due to the higher exit velocities experienced with the smaller diameters, and consequentially, a relatively uniform heat transfer distribution less susceptible to crossflow effects. The local distributions between models B and C differ only slightly due to the low jet to jet spanwise interaction occurring at these relatively high spanwise pitch values.

The streamwise resolved heat transfer indicated that the crossflow had a significant impact on the overall average heat transfer for the large diameter vane. At $H/D=1.58$ and $G=4.5 \text{ kg/m}^2\text{s}$, a decrease of 50% is seen in the average streamwise Nusselt number from the leading edge to the last row, this decrease is only 23% with the smaller jet diameter vanes. A significant amount of the heat transfer enhancement when using the smaller diameter jets occurs in the stagnation region. What adds to this enhancement is the lack of jet deflection for these smaller jet diameters, rendering the heat transfer

relatively steady under the jets. For the larger diameter jets, the distribution dissipates as the crossflow builds up across the array, thus reducing the overall average heat transfer.

All models obtained a decreasing average heat transfer with an increase in jet to target spacing, yielding the highest heat transfer at $H/D=1.58$. At a constant mass flow, the use of micro-jets enhanced the overall average heat transfer coefficient by 63%, but in order to attain such a mass flow a 75% increase in pressure drop is required (relative to larger diameter vane). Conversely, if the pressure drop percentage is fixed, the micro-jet diameter vane will still yield an enhancement in the heat transfer coefficient of 34.3%²⁵. Crude correlations were derived to obtain the required mass flow for a desired average heat transfer. The analysis revealed that for a desired average heat transfer coefficient, the use of micro-jets in the HPT vane would decrease the mass flow by 74%, relative to the larger diameter jets.

²⁵ at $\Delta P = 6\%$.

7.0 Conclusions and Recommendations for Future Work

Throughout the study, several conclusions were derived and are summarized below:

- A single axisymmetric semi-confined micro-jet of diameter 0.5mm was studied numerically at several Reynolds numbers and jet to target ratios. The analysis revealed that the K-epsilon RNG turbulence model obtained the closest agreement with experimental work both in terms of local and average heat transfer. An 8% decrease in heat transfer was obtained when increasing the jet to target spacing from 1D to 6D, which is in accordance with observed experimental trends for larger nozzles. Flow field patterns indicated that toroidal vortices formed in the vicinity of the stagnation region, where their center moved radially outward with increasing Reynolds number and jet to target spacing. Overall, a good concurrence was achieved in predicting the general trends in heat transfer and hydrodynamics of a single jet at various parameters.
- A full HPT vane was modeled based on a realistic 2nd stage impingement only HPT vane. For validation purposes, a simplified vane was created by strategically positioning the impinging jets only in the flat section of the airfoil. Using two well known turbulence models, Kw SST and K-epsilon RNG, the results were compared to experimental correlations at two different jet to target ratios ($H/D=1.58$, $H/D=3$) and Reynolds numbers 5000, 15000. The deviations indicated that both turbulence models were capable of replicating the mean experimental trends quite well. The hydrodynamics of the array was captured with excellent agreement, and the average heat transfer was obtained at a minimum deviation of 7.8% using the Kw SST model. It was concluded that the Kw SST model yields better predictions when considering the flow of a three dimensional impinging array, and as such, is used for the full vane analysis.
- A full realistic vane was considered with impinging jets throughout the pressure and suction surfaces of the airfoil. The results indicated that the static pressure decreased across the array, the crossflow increased as downstream jets introduced air, and consequentially, the jet exit velocity increased across the array. Increasing the jet to target spacing from 1.58D to 3D decreased the crossflow ratio distribution, increased the overall average Reynolds number, but decreased the average heat transfer on the vane by 49%.

Analysis of the local distribution indicated that the major decrease with increasing jet to target spacing occurred in the stagnation region under the jets, but also due to a decrease in forced channel convection for the larger channel height, and lower crossflow velocities. The turbulence model was capable of capturing the crossflow effect on the local Nusselt number distribution, where it was observed that the distribution and heat transfer footprint were swept by the upstream spent air. Increasing the mass flow per unit of heated area consequentially increases the array Reynolds number, yielding correspondingly higher exit velocities and heat transfer. Conversely, increasing the mass flow had no effect on the crossflow distribution, which is in accordance with experimental trends.

- Two models were used to assesses smaller diameter impinging jets, with $D=0.25\text{mm}$. The results indicated that halving the diameter of the jets doubles the overall average Reynolds number across the array, and reduces the increasing Reynolds number gradient across the array. A significant decrease in crossflow ratio occurs with the smaller diameters. This is attributed to the increase in non-dimensional spanwise spacing which occurs when D is halved. Due to the lower crossflow and higher exit velocities of the smaller jets, the penetration into the crossflow is much higher; as such, the local heat transfer distribution is steadier.
- For all models solved, an increase in jet to target spacing ($1.58D \rightarrow 3D$) decreased the overall heat transfer rate on the vane. As such, it is recommended that the jet to target spacing remain as small as possible.
- At a constant mass flow, the use of micro-jets enhanced the overall average heat transfer coefficient by 63%, but, the pressure drop percentage to obtain the same mass flow with the smaller jets is significantly higher, 75%. Conversely, if it is desired to fix the pressure drop percentage across the vane, instead of the mass flow, the smaller diameters will still yield an enhancement of 34.3% in average heat transfer rate. Crude correlations revealed that for a desired average heat transfer coefficient, the use of micro-jets in the HPT vane would decrease the mass flow per unit of heated area by 74%, relative to the larger diameter jets.

Overall, the study was a success. A thorough analysis of numerical modeling of impinging jets was completed. A total of 12 different geometries were studied, encompassing a single axisymmetric jet to a full analysis of an impinging array within a curved airfoil. The study revealed that commercially available turbulence models can attain a reasonable agreement with experimental work, and that micro-jets for use in gas turbine blades have tremendous potential in increasing heat transfer. Nevertheless, the results must be taken carefully and at the most, as a preliminary indication of the benefits of micro-jets. To solidify the analysis, the author recommends the following future work:

- The results indicated a decreasing heat transfer with an increasing jet to target spacing. It is possible that an optimal point lies between $H/D=1.58$ and $3D$, or conversely that a higher heat transfer can be obtained at an even lower jet to target spacing (i.e. $H/D=1$). As such, an extended analysis of several jet to target ratios for a full vane could aid in obtaining a realistic optimal point for maximum heat transfer.
- A micro-jet array should be analyzed in comparison to a larger diameter vane while keeping the same open area ratio, A_f . This would greatly impact the pressure loss and heat transfer characteristic of the array, but would be necessary since a small A_f ratio is quite susceptible to plugging of the holes.
- Using a flat impinging array, experimental work should analyze the effect of micro-jets while keeping the geometry of the array constant. These results would aid in confirming the numerical trends obtained herein.

Appendix

A1: Crossflow to Jet Mass Velocity Ratio, G_c/G_j , Extraction and Calculation

Figure A1 – Validation vane $H/D=1.58$ illustrating cross-flow mid planes

The crossflow mass velocity is extracted via the mid planes shown in Figure A1. The mass flow going through them is divided by the constant cross-sectional area, yielding G_c . To obtain the G_c for a specific row number, two planes are averaged. For example, for the G_c of row 2, the rightmost plane and the plane downstream from it are averaged, basically averaging the crossflow half a pitch upstream and downstream from that jet row. This is the convention use in the experimental work of Florschuetz et al [31], Van Treuren et al [28] and Bailey and Bunker [33]. The jet mass velocity, G_j , for a specific row is simply the average jet exit mass flow divided by its cross-sectional area (the mass flow for all three spanwise jets is not added, but averaged).

A2: Discharge Coefficient

The discharge coefficient for a nozzle is the ratio of the actual and ideal exit mass flow through the orifice. The ideal mass flow is obtained from isentropic relations at the same pressures to that of the actual nozzle. More specifically, the ideal jet mass velocity through an orifice can be calculated from [34]:

$$G_{Ideal} = P_o \left(\frac{P}{P_o} \right)^{1/\gamma} \left\{ \frac{2\gamma}{\gamma-1} \left(\frac{1 - \left(\frac{P}{P_o} \right)^{\frac{\gamma-1}{\gamma}}}{RT_o} \right) \right\}^{1/2} \quad (A.1)$$

Where the P and P_o are the absolute static pressure at the exit and inlet of the nozzles, respectively. T_o is the jet temperature (500°K), γ is the ratio of specific heats, and R the characteristic gas constant of air. The pressure values are averaged across the cross-sectional exit/inlet area of each of the orifices, and the ideal jet mass velocity is then calculated for each individual jet. The discharge coefficient for each jet is calculated via:

$$C_d = \frac{G_{Actual}}{G_{Ideal}} \quad (A.2)$$

Where the actual jet mass velocity, G_{Actual} , is simple extracted from Fluent (exit mass flow / cross-sectional jet area). The discharge coefficient is averaged for the entire array, and can then be used for calculating the Florschuetz et al [31] G_o/G_j distribution (Eq. 2.12)

A3: Katti and Prabhu [24] Streamwise Resolved Correlation Coefficients

The heat transfer correlation for the streamwise resolved NU is given again for completeness:

$$Nu = a_o (Re^{a_1} Pr^{0.33}) \left(\frac{G_j}{\bar{G}_j} \right)^{a_2} \left(1 - b_o \frac{G_c}{G_j} \right)^{b_1} \quad (A.3)$$

Where the coefficients can be obtained from Table A.1:

Table A.1 – Katti and Prabhu [34] Eq. A.3 coefficients

Configuration			a0	a1	a2	b0	b1
5,	2,	1	0.11	0.74	0.4	0.21	3.5
5,	2,	2	0.09	0.74	0.4	0.27	3.5
5,	2,	3	0.09	0.74	0.8	0.5	2.6
5,	4,	1	0.115	0.71	0.8	0.24	2.4
5,	4,	2	0.043	0.81	0.9	0.27	2.4
5,	4,	3	0.052	0.78	0.9	0.32	2.6
5,	6,	1	0.02	0.9	1	0.36	2
5,	6,	2	0.033	0.83	1	0.3	1.8
5,	6,	3	0.049	0.78	1	0.5	2

The configuration is represented by (X_n/D , Y_n/D , Z/D) or (streamwise spacing, spanwise spacing, jet to target spacing). The closest configuration to the validation model is highlighted in bold: 5, 4, 3, where the only difference is the streamwise spacing of 5D, instead of 4D for the validation model.

A4: Segmented Airfoil Wall & the Calculation of the Average Streamwise Resolved Heat Transfer

Figure A2 – Effective average area for each jet – validation model

Figure A2 illustrates two rows of jets and their corresponding affected area, for which the heat transfer coefficient is averaged. The average streamwise heat transfer for a given row is the average of each jet in that spanwise row (in Figure A2, across the span). This is the convention established in most experimental work [28-35], and thus to be as consistent as possible, it was replicated in the validation vane. The average heat transfer for a given jet is therefore calculated from a square patch of length half a pitch in all directions. The complexity arises due to the curved surface of the vane, which makes it difficult for Fluent to extract the heat transfer for each square patch. Fluent does not have the capability to form curved planes, only straight, and therefore, the airfoil surface had to be segmented prior to meshing the model. Figure A3 illustrates the segmented airfoil surface, where each patch was colored differently for clarity:

Figure A3 – Segmented Airfoil Surface, validation model, $H/D=1.58$

Each segment is defined as a wall, and is given the same temperature boundary conditions once imported into fluent. It is then possible to extract the average heat transfer for each jet and plot the streamwise resolved heat transfer distribution.

A5: Complete HPT Vane Mesh

The figures below illustrate the mesh for the complete base HPT vane: $H/D=1.58$, $D=0.51\text{mm}$.

Figure A4 – HPT base vane mesh, $H/D=1.58$, $D=0.51\text{mm}$

A closer look at the boundary layer region is given below:

Figure A5- HPT base vane mesh, boundary layer, $H/D=1.58$, $D=0.51\text{mm}$

A6: Crossflow Planes for Full Vane Geometry

The extraction of the G_c/G_j ratio requires mid planes located between the jets. For the validation vane (see A1), the mid planes are located at equal distances from each other, as the streamwise spacing is constant. For the full vane geometry, this is not the case. The procedure to extract the data is exactly the same as the validation vane (see A1), the locations are the only difference. Figure A6 illustrates the position of the crossflow mid planes:

Figure A6 – Crossflow mid planes for G_c/G_j extraction, full HPT vane, $H/D=1.58$, $D=0.51\text{mm}$

A7: Mesh sensitivity for HPT Vane models B and C

A formal sensitivity study was accomplished for the base vane and the validation, as such, only the local distribution was checked for consistency with an increase in refinement. Since the resolution was a ready set for the base vane, and the only change was a decrease in diameter for models B and C (which required a finer mesh in certain locations), two grids were compared: "Base", which had a close grids resolution to the verified one from model A, and "fine" which increased the resolution. Figures A7, and A8, illustrate the local heat transfer distribution on the pressure surface for model B and C, respectively. The agreement is excellent, proving yet again, mesh independence.

Figure A7 – Local heat transfer distribution on pressure surface for two grids, model B, $H/D=1.58$, $D=0.25\text{mm}$

Figure A8 - Local heat transfer distribution on pressure surface for two grids, model C, $H/D=1.58$, $D=0.25\text{mm}$

A8: Streamwise Resolved Heat Transfer for Full HPT Vane Geometry

For the validation vane, the streamwise resolved heat transfer was easily calculated as the streamwise spacing remained constant, and thus the patch under the jet had a constant area (See A4). For the full vane geometry, a new convention had to be used. The streamwise resolved heat transfer for an integer row (i.e. 1, 2, 3) is simply the average heat transfer for a 1D spanwise strip at that jet row location. Non-integer rows (i.e. 1.5, 2.5, 3.5) represent the average heat transfer for the in-between regions, 1.5 being in between jets 1 and 2. Figure A9 illustrates the row number designations for full vane geometry:

Figure A9 – Streamwise row numbers for full vane geometry

The airfoil surface is segmented as shown above prior to meshing. Upon convergence, average heat transfer values can be obtained for each of the surfaces above, yielding the streamwise resolved heat transfer distribution for full vane geometry.

A9: Local Nusselt Number Extraction

The local distribution for the validation and full HPT vane models is defined in this study as the average spanwise heat transfer rate of a 1 diameter strip cutting through the middle column of jets. The resolution was chosen due to the appropriate dimension of the flow problem, which can be defined as the diameter of the impinging jet. Likewise, single jet results indicate that at $r/D=1$ only the stagnation region is obtained, which can be considered local. A smaller resolution of $0.5D$ was investigated and no major significant differences were obtained. Figure 4.44 illustrates the 1D strip which wraps around the airfoil surface. The airfoil vane surface is segmented prior to meshing for average heat transfer purposes; as such the 1d strip could not be included prior to meshing as well. Therefore, a program was coded via VBA and Excel to extract the local data from the airfoil vane, and average only the values located between the coordinates of the 1D strip.

A brief breakdown of the code is given for completeness: The required input for the program is the heat transfer coefficient for each node and its x coordinate. Separately, a list of the heat transfer coefficient for each node and its z coordinate. This is accomplished via a journal file which extracts the heat transfer coefficient for x, and z separately from fluent²⁶. The files are read into Excel, and the code is initialized. The code matches corresponding x, z, and heat transfer values into pairs. It then isolates nodes which are located within the range of the 1D strip cutting through the middle column of jets. It proceeds to average the values and delete any duplicates which may be encountered (required due to the tetrahedral mesh).

Local Heat transfer extraction VBA code:

Sub ExtractLocal()

```
Dim rowNum As Integer
Dim heat As Double
Dim nheat As Double
Dim ndRow As Integer
Dim zValue As Double
Dim deleted As Integer
Dim xValue As Double
Dim tValues As Integer
Dim newHeat As Double
Dim nextValue As Double
```

²⁶ Fluent cannot export a given variable (heat transfer) with its x, y, and z coordinates. It must be done one at time.

```

Dim nextHeat As Double
Dim newX As Double
Dim model As String
Dim zStart As Double
Dim zEnd As Double
Dim diam As Double

```

```

'model =A --> S-B-XXX-12'
'model =B --> S-S-XXX-24'
'model =C --> S-S-XXX-4D'

```

```

'CHOOSE MODEL****'

```

```

model = "A"

```

```

'*****'

```

```

If model = "A" Then
    diam = 0.51
    zStart = 5.5 * diam
End If

```

```

If model = "B" Or model = "C" Then
    diam = 0.25
    zStart = 11.74 * diam
End If

```

```

zEnd = zStart + diam

```

```

Sheets("Sheet1").Select
If Cells(3, 4).Value = 1 Then
    Application.ScreenUpdating = True
Else
    Application.ScreenUpdating = False
End If

```

```

totRows = Cells(1, 2).Value

```

```

**** finding x, z partners in matrix values

```

```

For rowNum = 5 To totRows

```

```

    heat = Cells(rowNum, 2).Value
    Cells(rowNum, 6).Value = Cells(rowNum, 1)
    Cells(rowNum, 8).Value = heat

```

```

    For ndRow = 5 To totRows

```



```
nheat = Cells(ndRow, 4).Value  
checks = heat / nheat
```

```
If checks = 1 Then  
    zValue = Cells(ndRow, 3).Value  
    Cells(rowNum, 7).Value = zValue  
End If
```

```
Next ndRow  
Next rowNum
```

******Deleting all zeros from the matrix**

```
Columns("F:H").Select  
Selection.Copy  
Sheets("Sheet2").Select  
Columns("A:C").Select  
ActiveSheet.Paste  
Application.CutCopyMode = False
```

```
deleted = 0
```

```
For rowNum = 5 To totRows
```

```
    heat = Cells(rowNum, 2).Value
```

```
    If heat = 0 Then  
        Rows(rowNum).Select  
        Selection.Delete Shift:=xlUp  
        deleted = deleted + 1  
    End If
```

```
Next rowNum
```

```
totRows = totRows - (deleted / 2)  
Cells(1, 1) = totRows
```

' * copying back into main sheet - clearing contents**

```
Sheets("Sheet1").Select  
Columns("F:H").Select  
Selection.ClearContents  
Sheets("Sheet2").Select  
Columns("A:C").Select  
Selection.Copy  
Sheets("Sheet1").Select
```

```
Columns("F:H").Select
ActiveSheet.Paste
Application.CutCopyMode = False
Sheets("Sheet2").Select
Columns("A:C").Select
Selection.ClearContents
Sheets("Sheet1").Select
Cells(1, 1).Select
```

```
For rowNum = 5 To totRows
```

```
    xValue = 1000 * Abs(Cells(rowNum, 6).Value)
    zValue = 1000 * Abs(Cells(rowNum, 7).Value)
```

```
    Cells(rowNum, 6).Value = xValue
    Cells(rowNum, 7).Value = zValue
```

```
Next rowNum
```

```
Cells(4, 6) = "X"
Cells(4, 7) = "Z"
Cells(4, 8) = "Heat Flux"
```

```
'*****sort Z values
```

```
Range("F4:H4").Select
Selection.AutoFilter
ActiveWorkbook.Worksheets("Sheet1").AutoFilter.Sort.SortFields.Clear
ActiveWorkbook.Worksheets("Sheet1").AutoFilter.Sort.SortFields.Add Key:=Range _
("G4"), SortOn:=xlSortOnValues, Order:=xlAscending, DataOption:=xlSortNormal
With ActiveWorkbook.Worksheets("Sheet1").AutoFilter.Sort
    .Header = xlYes
    .MatchCase = False
    .Orientation = xlTopToBottom
    .SortMethod = xlPinYin
    .Apply
End With
```

```
****find values located within a 1D spanwise distance from jet centerline
```

```
tValues = 0
```

```
For rowNum = 5 To totRows
```

```
    zValue = Cells(rowNum, 7).Value
    xValue = Cells(rowNum, 6).Value
    heat = Cells(rowNum, 8).Value
```


If zValue > zStart And zValue < zEnd Then '1d strip for desired column'

tValues = tValues + 1

Cells(tValues + 4, 11).Value = zValue

Cells(tValues + 4, 10).Value = xValue

Cells(tValues + 4, 12).Value = heat

End If

Next rowNum

totRows = tValues + 4

*****copy new values**

Columns("F:H").Select

Selection.ClearContents

Columns("J:L").Select

Selection.Copy

Columns("F:H").Select

ActiveSheet.Paste

Application.CutCopyMode = False

Columns("J:L").Select

Selection.ClearContents

Cells(4, 6) = "X"

Cells(4, 7) = "Z"

Cells(4, 8) = "Heat Flux"

Range("F4:H4").Select

Selection.AutoFilter

ActiveWorkbook.Worksheets("Sheet1").AutoFilter.Sort.SortFields.Clear

ActiveWorkbook.Worksheets("Sheet1").AutoFilter.Sort.SortFields.Add Key:=Range _
("F4"), SortOn:=xlSortOnValues, Order:=xlDescending, DataOption:=xlSortNormal

With ActiveWorkbook.Worksheets("Sheet1").AutoFilter.Sort

.Header = xlYes

.MatchCase = False

.Orientation = xlTopToBottom

.SortMethod = xlPinYin

.Apply

End With

*****average same or similar X values across the span**

Columns("F:H").Select

Selection.Copy

Sheets("Sheet2").Select

Columns("A:C").Select

```

ActiveSheet.Paste
Application.CutCopyMode = False
Range("A4").Select
Selection.Subtotal GroupBy:=1, Function:=xlAverage, TotalList:=Array(3), _
Replace:=True, PageBreaks:=False, SummaryBelowData:=True
Range("F9").Select
ActiveSheet.Outline.ShowLevels RowLevels:=2
Columns("B:B").Select
Selection.Delete Shift:=xlToLeft
Cells.Replace What:=" Average", Replacement:="", LookAt:=xlPart, _
SearchOrder:=xlByRows, MatchCase:=False, SearchFormat:=False, ReplaceFormat:=False

```

***** copy values back to main sheet**

```

Columns("A:B").Select
Selection.SpecialCells(xlCellTypeVisible).Select
Selection.Copy
Sheets("Sheet1").Select
Columns("I:J").Select
ActiveSheet.Paste

```

```

Sheets("Sheet2").Select
Columns("A:B").Select
Selection.ClearContents
Selection.ClearOutline

```

```

Sheets("Sheet1").Select
Columns("F:H").Select
Selection.ClearContents
Columns("I:J").Select
Selection.Cut Destination:=Columns("F:G")
Columns("F:G").Select

```

```

Set dest = Sheets("Sheet1").Range("F:F")
totRows = Application.WorksheetFunction.CountA(dest) + 2

```

```

Cells(totRows + 1, 6).Value = " "
Cells(totRows + 1, 7).Value = " "

```

*****Averaging values**

```

deviation = Cells(1, 4)
counter = 0

```

```

For rowNum = 5 To totRows - 1

```



```
curValue = Cells(rowNum, 6).Value
curHeat = Cells(rowNum, 7).Value
nextValue = Cells(rowNum + 1, 6).Value
nextHeat = Cells(rowNum + 1, 7).Value
check = nextValue / curValue
Cells(rowNum, 8).Value = check
```

```
If check > deviation Then
    counter = counter + 1
Else
    counter = 0
End If
```

```
Cells(rowNum, 9).Value = counter
Next rowNum
```

```
For rowNum = 5 To totRows
    counter = Cells(rowNum, 9).Value
    ncounter = Cells(rowNum + 1, 9).Value
```

```
If ncounter < counter Then
    Cells(rowNum + 1, 10).Value = "LAST"
End If
```

```
Next rowNum
```

```
For rowNum = 5 To totRows
```

```
checkie = Cells(rowNum, 9).Value
pcheckie = Cells(rowNum - 1, 9).Value
```

```
If checkie = 0 And pcheckie < 1 Then
    Cells(rowNum, 11).Value = Cells(rowNum, 6).Value
    Cells(rowNum, 12).Value = Cells(rowNum, 7).Value
```

```
End If
```

```
If Cells(rowNum, 10).Value = "LAST" Then
```

```
    counter = Cells(rowNum - 1, 9).Value
```

```
    xValue = 0
```

```
    curHeat = 0
```

```
    For similar = 0 To counter
```

```
        xValue = xValue + Cells(rowNum - counter + similar, 6).Value
        curHeat = curHeat + Cells(rowNum - counter + similar, 7).Value
```

Next similar

```
Cells(rowNum, 11).Value = xValue / (counter + 1)  
Cells(rowNum, 12).Value = curHeat / (counter + 1)
```

End If

Next rowNum

*****copy new values into sheet and delete blank cells**

```
Columns("K:L").Select  
Selection.Copy  
Sheets("Sheet2").Select  
Columns("A:B").Select  
ActiveSheet.Paste  
Application.CutCopyMode = False
```

```
For rowNum = 5 To totRows  
  If IsEmpty(Cells(rowNum, 1)) Then  
    Cells(rowNum, 1).Value = "blank"  
  End If  
Next rowNum
```

```
For rowNum = 5 To totRows  
  If Cells(rowNum, 1).Value = "blank" Then  
    Rows(rowNum).Select  
    Selection.Delete Shift:=xlUp  
    rowNum = rowNum - 1  
  End If  
Next rowNum
```

```
Set dest = Sheets("Sheet2").Range("A:A")  
tValues = Application.WorksheetFunction.CountA(dest)  
totRows = tValues + 4
```

***** deleting duplicated entries**

```
Sheets("Sheet1").Select  
deviation = Cells(2, 4).Value  
Sheets("Sheet2").Select  
tValues = 0
```

```
For rowNum = 5 To totRows
```



```
curValue = Cells(rowNum, 1).Value
curHeat = Cells(rowNum, 2).Value
nextValue = Cells(rowNum + 1, 1).Value
nextHeat = Cells(rowNum + 1, 2).Value
```

```
check = nextValue / curValue
Cells(rowNum, 3).Value = check
```

```
If check > deviation Then
```

```
Cells(rowNum, 4).Value = "delete"
Cells(rowNum + 1, 4).Value = "delete"
```

```
posNeg = curHeat - nextHeat
Cells(rowNum, 5).Value = posNeg
```

```
If posNeg < 0 Then
```

```
newHeat = Cells(rowNum + 1, 2).Value
newX = Cells(rowNum + 1, 1).Value
Cells(rowNum + 1, 7).Value = newHeat
Cells(rowNum + 1, 6).Value = newX
```

```
Else
```

```
newHeat = Cells(rowNum, 2).Value
newX = Cells(rowNum, 1).Value
Cells(rowNum, 7).Value = newHeat
Cells(rowNum, 6).Value = newX
```

```
End If
```

```
Else
```

```
If Cells(rowNum, 4).Value <> "delete" And Cells(rowNum + 1, 4) <> "delete" Then
```

```
Cells(rowNum, 6).Value = curValue
Cells(rowNum, 7).Value = curHeat
```

```
End If
```

```
End If
```

```
Next rowNum
```

```
'delete empty cells
```

```
For rowNum = 5 To totRows
```

```
If IsEmpty(Cells(rowNum, 6)) Then
Cells(rowNum, 6).Value = "blank"
```

```
End If
```

Next rowNum

For rowNum = 5 To totRows

 If Cells(rowNum, 6).Value = "blank" Then

 Rows(rowNum).Select

 Selection.Delete Shift:=xlUp

 rowNum = rowNum - 1

 End If

Next rowNum

Set dest = Sheets("Sheet2").Range("G:G")

tValues = Application.WorksheetFunction.CountA(dest)

totRows = tValues + 4

' *****copying back to main page - clearing sheet – Final Values**

Columns("A:E").Select

Selection.ClearContents

Range("I11").Select

Sheets("Sheet1").Select

Columns("F:L").Select

Selection.ClearContents

Range("F7").Select

Sheets("Sheet2").Select

Columns("F:G").Select

Selection.Copy

Sheets("Sheet1").Select

Columns("F:G").Select

ActiveSheet.Paste

Application.CutCopyMode = False

Sheets("Sheet2").Select

Columns("F:G").Select

Selection.ClearContents

Sheets("Sheet1").Select

Cells(4, 6) = "X"

Cells(4, 7) = "Heat Flux"

Range("F4:G4").Select

Selection.Font.Bold = True

 With Selection

 .HorizontalAlignment = xlCenter

 .VerticalAlignment = xlBottom

 .WrapText = False

 .Orientation = 0

 .AddIndent = False

 .IndentLevel = 0

 .ShrinkToFit = False


```
.ReadingOrder = xlContext  
.MergeCells = False  
End With
```

```
Cells(1, 1).Select
```

```
End Sub
```

References

- [1] Tamarin, Y. Protective Coatings for Turbine Blades. 1st. ASM International, 2001
- [2] Han, Je-Chin. Gas Turbine Heat Transfer and Cooling Technology. 1st. Taylor & Francis, 2001
- [3] Koff, Bernard, 2004, "Gas Turbine Technology Evolution : A Designer's Perspective", Journal of Propulsion and Power, 20(4), pp. 577-595.
- [4] Jordal, Kristin, 2001, " Modeling and Performance of Gas Turbine Cycles with Various Means of Blade Cooling" Doctoral Thesis, Lund University, Sweden.
- [5] Haubert, R.C, Hsia. E, Maclin. M, Noe. M, and Brooks. R, 1980, "High Pressure Turbine Blade Life Sensitivity", AIAA/SAE/ASME 16th Joint Propulsion Conference.
- [6] Lefebvre, Arthur H. Gas Turbine Combustion. 2nd. Philadelphia: Taylor & Francis, 1999.
- [7] Siegel, Robert and Spuckler, Charles, 1998, "Analysis of thermal radiation effects on temperatures in turbine engine thermal barrier coatings", Material Sciences and Engineering, A245, pp.150-159
- [8] Halila , E.E, D. T. Lenahan, and T.T. Thomas. "Energy efficient engine: High Pressure turbine test hardware detailed design report." NASA - NAS3-0643 (1982): 1-194.
- [9] Gao, Xiufang and Bengt Sundén, 2003. "Experimental Investigation of the Heat Transfer Characteristics of Confined Impinging Slot Jets", Experimental Heat Transfer, 16, pp. 1-18
- [10] Dano, Bertrand, James Liburdy, and Koonlaya Kanokjaruvijit, 2005. "Flow characteristics and heat transfer performances of a semiconfined impinging array of jets: effect of nozzle geometry." International Journal of Heat and Mass Transfer, 48, pp. 691-701
- [11] Obot, N.T, A. S. Majumdar, and W.J.M. Douglas , 1979, " The Effect of Nozzle Geometry on Impingement Heat Transfer under a Round Turbulent jet", ASME proceedings Annual meeting, pp.1-12
- [12] Chambers, Andrew, David R.H. Gillespie, and Peter T. Ireland, 2006. "Enhancement of Impingement Cooling in a High Cross Flow Channel using Shaped Impingement Cooling Holes", Proceedings of the ASME Turbo EXPO 2005, pp.995-1004
- [13] Riahi, Ardeshir and Luis A. Tapia, 2006. " Numerical Investigation of the Impact of Hole Geometry on the Impingement Heat Transfer Characteristics", Annual Forum - American Helicopter Society, pp.2083-2091
- [14] Garimella, S.V, and R.A. Rice, 1995. "Confined and Submerged Liquid Jet Impingement Heat Transfer", Journal of Heat Transfer, 117, pp. 871-877
- [15] Martin, H, 1977. "Heat and mass transfer between impinging gas jets and solid surfaces." Advances in heat transfer, 13, pp. 1-60

- [16] Gardon, Robert, and J. Cahit Akfirat, 1965. "The Role of Turbulence in Determining The Heat-Transfer Characteristics of Impinging Jets", *International Journal of Heat and Mass Transfer*, 8, pp. 1261-1272
- [17] Jambunathan, K.E. Lai, M. a. Moss and B.L. Button, 1992. "A review of heat transfer data for single circular jet impingement." *International Journal of Heat and Fluid Flow*, 13, pp. 106-115.
- [18] Huang, Lianmin and Mogammed El-Genk, 1994. "Heat Transfer of an Impinging Jet on a Flat Surface", *International Journal of Heat and Mass Transfer*, 37(13), pp. 1975-1923
- [19] Hoffman, Herbert, Matthias kind, and Holger Martin, 2007. "Measurements on steady state heat transfer and flow structure and new correlations for heat and mass transfer in submerged impinging jets"
- [20] R.J Goldstein and A.I Behbahani, 1982, Impingement of a circular jet with and without cross flow, *Journal of heat and mass transfer*, Vol 25, p 1377-1382.
- [21] Gardon, R, and Carbonpue,J. "Heat transfer between a flat plate and jets of air impinging on it." *International developments in heat transfer* 2(1962): 454-460.
- [22] Saad, N.R, W.J.M Douglas, and A.S. Mujumdar, 1977. "Prediction of Heat Transfer under an Axisymmetric Laminar Impinging Jet", *Industrial and Engineering Chemistry Fundamentals*, 16(1), pp. 148-154
- [23] Obot, N.Y, Mujumdar, A.S, and Douglas, W.J. "Effect of semi-confinement on impingement heat transfer." *Proc. 7th international heat transfer conference* 3(1982): 395-400.
- [24] Behnia, M, S. Parneix, Y. Shabany, and P.A. Durbin. "Numerical study of turbulent heat transfer in confined and unconfined impinging jets." *International journal of heat and fluids flow*. 20(1999): 1-9.
- [25] Fitzgerald, Janice, and Suresh Garimella, 1998. " A study of the flow field of a confined and submerged impinging jet" *International Journal of heat and mass transfer*, 41 , pp. 1025-1034.
- [26] Li, Xianchang, J.Leon Gaddis, and Ting Wang, 2005, "Multiple flow patterns and heat transfer in confined jet impingement" *International Journal of Heat and Fluid Flow*, 26, pp. 746-754
- [27] Koseoglu, M.F, and S.Baskaya, 2008. " The effect of flow field and turbulence on heat transfer characteristics of confined circular and elliptic impinging jets", *International Journal of Thermal Sciences*, 47, pp. 1332-1346.
- [28] Van Treuren, K., 1994, PhD Thesis: Impingement Flow Heat Transfer Measurements of Turbine Blades Using a Jet Array, University of Oxford, pp. 1-269 .
- [29] Van Treuren, K, Wang, Z., Ireland, P., and Jones, I., and Kohler, S., 1996, "Comparison and Prediction of Local and Average Heat Transfer Coefficients under an Array of Inline and Staggered Impinging Jets", 96-GT-163, IGTI Congress and Exhibition, Birmingham, UK.

- [30] Kercher,D and W. Tabakoff, 1970 " Heat transfer by a square array of round air jets impinging perpendicular to a flat surface including the effect of spent air", Journal of engineering for power, 92, pp. 73-82.
- [31] Florschuetz, L.W., Metzger, D.E., and Truman, C.R., Jet array Impingement with Crossflow – Correlation of Streamwise Resolved Flow and Heat Transfer Distributions, NASA Contractor report 3373, Department of Mechanical Engineering, Arizona State University, Tempe, January 1981.
- [32] Florschuetz, L.W., Truman C.R., and Metzger, D.E., 1980, "Periodic streamwise variations of heat transfer coefficients for jet array impingement with crossflow", ASME Journal of heat transfer, 103, pp. 337-342.
- [33] Bailey, J.C., and R.S. Bunker,2002, "Local heat transfer and flow distributions for impinging jet arrays of dense and sparse extent", Proceedings of the ASME Turbo Expo 2002.
- [34] Katti, Vadiraj, and S.V. Prabhu, 2008 "Influence of Spanwise Pitch on Local Heat Transfer for Multiple Jets with Crossflow", Journal of Thermophysics and Heat Transfer, 22(4), pp.654-668
- [35] Brevet, P, C. Dejeu, E.Dorignac, M.Jolly,and J.J. Vullierme, 2002, " Heat transfer to a row of impinging jets in consideration of optimization", International Journal of Heat and Mass Transfer, 45, pp.4191-4200
- [36] Zuckerman, Neil, and Noam Lior, 2005 "Impingement Heat Transfer: Correlations and Numerical Modeling", Transactions of the ASME, 127 , pp.544-552
- [37] Huang, Yizhe, Srinath V.Ekkad, and Je-Chin Han, 1998, " Detailed Heat Transfer distributions under an Array of Orthogonal Impinging Jets", Journal of Thermophysics and Heat Transfer, 12(1), pp.73-79
- [38] Gao, Lujia., 2003, Masc. Thesis: Effect of Jet Hole Arrays Arrangement on Impingement Heat Transfer, Louisiana State University, p 1-73 .
- [39] Gao, Lujia, Srinath V. Ekkad, and Ronald S. Bunke, 2005 "Impingement Heat Transfer, Part I: Linearly Stretched Arrays of Holes" Journal Of Thermophysics and Heat Transfer, 19(1), pp.57-65
- [40] Andrews,G.E, J.Durance, C.I. Hussain, S.N. Ojobor,1987 "Full Coverage Impingement Heat TRansfer: Influence of the Number of Holes", Journal Of turbomachinery, 109, pp.557-563
- [41] Garimella,Suresh, and Boris Nenaydykh, 1996 "Nozzle-geometry effects in liquid jet impingement heat transfer", International Journal of heat and Mass transfer, 39(4), pp.2915-2923
- [42] Fabbri, Matteo, Shanjuan Jiang, and Vijay K. Dhir., 2005, "A Comparative Study of Cooling Of High Power Density Electronics Using Sprays and Microjets." Transactions of the ASME, 127, pp. 38- 48.
- [43] Phalnikar, K.A, F.S Alvi, and C.Shih., 2001, "Behavior of Free and Impinging Supersonic Microjets.", AIAA, pp. 1-15.

- [44] Leland, John E, Rengasamy Ponnappan, and Kevin S. Klasing., 2002, "Experimental Investigation of an Air Microjet Array Impingement Cooling Device." *Journal of Thermophysics and Heat Transfer*, 16, pp. 187-192.
- [45] Coussirat, M, J. Van beeck, M mestres, E egusguiza, J buchlin, and X escaler "Computational fluids dynamics modeling of impinging gas-jet systems:1, Assessment of eddy viscosity models." *Journal of fluids engineering* 127(2005): 691-703.
- [46] S. Spring and B. Weigand, 2006, CFD heat transfer predictions of a single circular jet impinging with Crossflow, AIAA/ASME Joint thermo-physics and heat transfer conference, p 1-11
- [47] Zhang, Yan, Jing-Yu Fan, and Johan Liu, 2005, "Numerical Investigation Based on CFD for Air impingement Heat Transfer in Electronics Cooling", *Proceedings of the Seventh IEEE CPMT Conference on High Density Microsystem Design and Packaging and Components Failure Analysis*, pp. 1-5
- [48] Shen, Jun, Moen Alyaser, and Abdlmomem H. Beitelmal, 2006 "Turbulent Heat transfer study of Inclined Impinging jets", *Proceedings of the AIAA/ASME Joint Thermophysics and Heat Transfer conference*, pp. 1-14
- [49] Kumar, B.V.N Rama and B.V.S.S.S. Prasad, 2008, "Computational flow and heat transfer of a row of circular jets impinging on a concave surface", *Heat mass transfer*, 44, pp. 667-678
- [50] El-Gabry, Lamyaa A, and Deborah A .Kiminski, 2005, "Numerical Investigation of jet impingement with cross Flow – Comparison of Yang-Shih and Standard k-e turbulence Models", *Numerical heat transfer, A*, 441-469
- [51] Angioletti M, E. Nino, and G. Ruocoo, 2004 "CFD turbulent modeling of jet impingement and its validation by particle image velocimetry and mass transfer measurements", *International journal of thermal Sciences*, Vol 44, p 349-356.
- [52] Coussirat, M, J. Van beeck, M mestres, E egusguiza, J buchlin, and X escaler., 2005, "Computational fluids dynamics modeling of impinging gas-jet systems:2. Application to an Industrial Cooling System Device" *Journal of fluids engineering*, 127, pp. 691-703.
- [53] White, Frank M. *Fluid Mechanics*. 5th. Boston: McGraw-Hill, 2003.
- [54] Roache,P, 1998, "Verification and Validation in Computational Science and Engineering." Hermosa Publishers, Albuquerque, NM.
- [55] Roache, P, 1998, "A Method for Uniform Reporting of Grid Refinement Studies, Quantification of Uncertainty in Computational Fluid Dynamics", *FED (ASME)* , 20, pp. 109-120,199.
- [56] San, Jung-Yang, and Wen-Zheng Shiap, 2006 " Effects of jet plate size and plate spacing on the stagnation Nusselt number for a confined circular air jet impinging on a flat surface", *International Journal of Heat and Mass Transfer*, 49, pp. 3477-3486

[57] Pence, Deborah V, Paul A. Boeschoten, and James A. Liburdy., 2003, "Simulation of Compressible Micro-Scale Jet Impingement Heat Transfer." *Journal of Heat Transfer*, 125, pp. 447-453.

[58] Shadlesky, P. S., 1983 "Stagnation Point Heat Transfer for Jet Impingement to a Plane Surface" *AIAA J.* 21, pp. 1214-1215

[59] FLUENT Inc. (2003) *Fluent 6.1 Users Guide*. FLUENT Inc., Lebanon, NH

[60] Logan, Earl and Roy, Ramendra. *Handbook of Turbomachinery – Chapter 4: Selection of a Gas Turbine System*. 2nd. CRC, 2003.