THE INFLUENCE OF FILM COOLING AND INLET TEMPERATURE PROFILE ON HEAT TRANSFER FOR THE VANE ROW OF A 1-1/2 STAGE TRANSONIC HIGH-PRESSURE TURBINE

DISSERTATION

Presented in Partial Fulfillment of the Requirements for the Degree Doctor of Philosophy in the Graduate School of The Ohio State University

By

Harika Senem Kahveci, M.Sc.

Graduate Program in Mechanical Engineering

The Ohio State University

2010

Dissertation Committee:

Professor Michael G. Dunn, Advisor

Professor Charles W. Haldeman

Professor J. William Rich

Professor Mohammad Samimy
ABSTRACT

The current study focuses on determination of the local heat flux for the airfoil and endwall surfaces of the vane row of a fully-cooled turbine stage. The measurements that are essential to this study were performed at the Ohio State University Gas Turbine Laboratory using the Turbine Test Facility operating in blowdown mode. The full-scale rotating turbine stage used consists of a high-pressure vane, a high-pressure rotor, and a low-pressure vane. Temperature, pressure, and heat-flux measurements are obtained at the proper corrected engine design conditions, such as the Flow Function (FF), the corrected speed, the stage Pressure Ratio (PR), and the temperature ratios of gas to wall and gas to coolant. The measurements are repeated for different vane inlet temperature profiles and different vane cooling flows in order to establish an in-depth understanding of the influence of film cooling on local heat transfer, and thus on cooling effectiveness. Double-sided Kapton heat-flux gauges are used for heat-flux measurements at different span locations along the airfoil surfaces and along the inner endwall. Film cooling is managed via numerous cooling holes located on the inner and outer endwalls, at the airfoil leading edge with a showerhead arrangement, at numerous locations on the airfoil pressure and suction surfaces, and at the vane trailing edge, which results in a fully-cooled first stage vane.

This is a unique data set in that the measurements were performed not only at the design corrected conditions for a rotating turbine stage, but also for a fully-cooled vane environment. It is the first time that heat transfer data obtained in such an environment has become available for a fully-cooled vane endwall. The unique film-cooled endwall heat transfer data demonstrated in contour plots reveals insight to the complex flow behavior that is dominant in this region, which becomes even more complicated with the addition of coolant.

Addition of cooling resulted in notable reductions in heat transfer levels, but the percent variation in heat transfer caused by the temperature profiles were still comparable
to that observed in an un-cooled environment. The variations between the profiles and the cooling levels are found to be comparable on the airfoil surface, as well as to those observed between the spans. The differences between the cooling levels were more clearly observed on the airfoil pressure surface than the suction surface, and coolant had more effect in reducing heat transfer at the inner spans. At the endwall region, the profile effects are more significant than the cooling effects, resulting in larger differences in heat transfer levels. Within the range of coolant variation studied, an increase in the coolant mass flow served to smooth out the large gradients due to flow complexity in the endwall heat transfer rather than increasing the cooling effectiveness even further.

The combined trailing edge and outer endwall cooling results in significant reduction in heat transfer at all surfaces, in a growing fashion towards the trailing edge, and at the endwall exit, while the purge flow through the wheel-space cavity does not have an influence on the vane heat transfer. The reduction achieved by the vane outer cooling is comparable to the reduction obtained by the highest cooling level studied.

The hot streak inlet profiles were performed with different alignments at varying magnitudes. Alignment with vane leading edge lowers heat transfer compared to the alignment with mid-passage both at the mid-span suction surface and through the endwall passage, and increases it at the endwall exit, while the pressure surface is found to be insensitive to this switch. When the magnitude of the hot streak is increased, no observable difference is observed at the endwall.

A comparison of the current results with those obtained from a previous research program with the un-cooled version of the vane with the same geometry at similar non-dimensional experimental design operating conditions gives good comparison on the pressure surface and at the endwall, but significantly lower heat transfer on the airfoil suction surface as would be anticipated due to the ingestion occurring through the cooling holes filling the plenum and being ejected onto the suction surface.

The goal of this research was to establish an extensive database for typical engine hardware with a film-cooled first stage vane, which represents the foundation for future turbomachinery film cooling modeling and component heat transfer studies. Until this time, such a database was not available within the gas turbine industry.
DEDICATION

To my parents Neşe & Ali Rıza Kahveci
I owe all of my success to you.
ACKNOWLEDGMENTS

I consider myself very fortunate to have been the research assistant of my advisor Professor Michael G. Dunn. I would like to thank him for providing me with such a unique and an exciting research opportunity, and I am grateful for all his guidance and his valuable advice during my studies towards a doctoral degree. The experimental part of this work would not have been possible without him and his technical staff. I have enjoyed my time here as a member of the OSU Gas Turbine Laboratory family. I would like to thank Prof. Charles W. Haldeman for his technical assistance during every phase of data processing, and his time for many discussions we had all along the way. It has been a pleasure for me to have technical conversations with him, which were always accompanied with his great sense of humor. The contributions of Dr. Randy M. Mathison during the calibration step of experimental data are also appreciated. I am indebted to Prof. Charles W. Haldeman, Prof. J. William Rich, and Prof. Mohammad Samimy for serving in my dissertation committee.

I greatly acknowledge General Electric Aviation, which provided the financial support to this project via the University Strategic Alliance Program. I also greatly appreciate General Electric Energy for providing me with an educational leave of absence for the period I have pursued my doctoral study.

Finally, I would like to thank my parents Neşе and Ali Rıza Kahveci, my sister Dr. Nazlı E. Kahveci, and my friends, for their love and support throughout my education. My parents have always been great role models in my life. My father has been the real motivation for me in pursuing my Ph.D. work, and has always encouraged me to do my best. I am very much grateful for everything they have done, which made this dissertation a possibility.
VITA

September 25, 1979……..Born – State College, Pennsylvania

Summer 2000…………..Summer Intern
TUSAŞ Aerospace Industries, Inc. (TAI)
Research and Development Department
Ankara, Turkey

Summer 2001…………..Summer Intern
Roketsan Missile Industries
Engineering and Development Department
Ankara, Turkey

2002………………….B.S. in Aeronautical Engineering
Middle East Technical University
Ankara, Turkey

Summer 2003…………..University Turbine System Research Program (UTSR) Fellow
Parker Hannifin Corporation
Gas Turbine Fuel Systems Division
Mentor, Ohio

2002 - 2004…………..Research Assistant
The Pennsylvania State University
Turbomachinery Heat Transfer Laboratory
University Park, Pennsylvania

2004…………………...M.Sc. in Aerospace Engineering
The Pennsylvania State University
University Park, Pennsylvania

2004 - 2006…………..Aero Design Engineer
General Electric Energy
Power Generation Technology
Greenville, South Carolina

2006 – Present………..Graduate Research Associate
The Ohio State University
Gas Turbine Laboratory
Columbus, Ohio
PUBLICATIONS


FIELDS OF STUDY

Major Field: Mechanical Engineering
TABLE OF CONTENTS

ABSTRACT .......................................................................................................................... ii
DEDICATION ....................................................................................................................... iv
ACKNOWLEDGMENTS ..................................................................................................... v
VITA ................................................................................................................................... vi
LIST OF FIGURES ........................................................................................................... xi
LIST OF TABLES .............................................................................................................. xxi
NOMENCLATURE AND ABBREVIATIONS ................................................................. xxii

Chapter 1 INTRODUCTION ............................................................................................. 1
  1.1 Significance of the Problem ...................................................................................... 2
  1.2 Scope and Objectives of the Research ..................................................................... 3

Chapter 2 FILM COOLING FUNDAMENTALS .............................................................. 6

Chapter 3 LITERATURE REVIEW .................................................................................... 15
  3.1 Experimental Data Base ......................................................................................... 16
    3.1.1 Flat-Plate Experiments .................................................................................... 16
    3.1.2 Cascade Data Base ......................................................................................... 17
    3.1.3 Rotating Turbine Data Base ........................................................................... 20
      3.1.3.1 Rotating Rig Experiments Without Cooling ............................................. 22
      3.1.3.2 Rotating Rig Experiments With Cooling .................................................. 26
  3.2 Computational Work ............................................................................................... 29

Chapter 4 EXPERIMENTAL METHODOLOGY ........................................................... 34
LIST OF FIGURES

Figure 2.1 The Three-Dimensional Separation of a Boundary Layer Entering a Turbine Cascade [3].............................................................................................................................................. 6

Figure 4.1 A View of the Turbine Facility........................................................................................................... 35

Figure 4.2 Rear View of the Rig With Compressed Air Supplies and Cooling Circuits.. 37

Figure 4.3 Large Cooling Facility....................................................................................................................... 38

Figure 4.4 Combustor Emulator – Upstream View (on the Left) and From Another Angle (on the Right).................................................................................................................. 40

Figure 4.5 A Close View of Combustor Emulator................................................................................................. 41

Figure 4.6 Turbine Test Rig.................................................................................................................................. 43

Figure 4.7 Turbine Rig Assembly Showing Parts (Not to Scale).......................................................................... 44

Figure 4.8 Cooling Hole Geometry on the Vane Airfoil Surface (Not to Scale)............................... 45

Figure 4.9 Gauge Locations and Cooling Hole Geometry at the Inner Endwall (Not to Scale).................................................................................................................................................. 46

Figure 4.10 Pyrex Heat-Flux Gauges in Vane Angel Wing Cavity Area ................................. 47

Figure 4.11 Gauge Locations on the Vane Airfoil............................................................................................... 49

Figure 4.12 Heat-Flux Gauges at 90% and 50% Spans (Top), 15% and 5% Spans (Middle), at the Endwall (Bottom)......................................................................................................................... 50

Figure 4.13 Vane Instrumentation Locations....................................................................................................... 51

Figure 4.14 Cross-Section of Kapton Gauges [112]............................................................................................ 52
Figure 5.10 Typical Heat Flux Values and Repeatability ................................................................. 95
Figure 5.11 Typical Stanton Numbers and Repeatability ................................................................. 98
Figure 5.12 Uncertainty in Stanton Number Calculation ................................................................. 101
Figure 6.1 Inlet Temperature Profiles With No Cooling and Nominal Cooling ............................... 111
Figure 6.2 Heat Flux and Stanton Number Distributions on the Vane Airfoil Surface
With No Cooling ................................................................................................................................. 114
Figure 6.3 Heat Flux and Stanton Number Distributions on the Vane Airfoil Surface
With Nominal Cooling – Case A ......................................................................................................... 115
Figure 6.4 Heat Flux Distributions on the Vane Airfoil Surface With No Cooling and
Nominal Cooling (Case A) .................................................................................................................. 116
Figure 6.5 Stanton Number Distributions on the Vane Airfoil Surface With No
Cooling and Nominal Cooling (Case A) ........................................................................................... 117
Figure 6.6 Heat Flux and Stanton Number Distributions at the Inner Endwall With No
Cooling .............................................................................................................................................. 120
Figure 6.7 Heat Flux and Stanton Number Distributions at the Inner Endwall With
Nominal Cooling – Case A ................................................................................................................... 121
Figure 6.8 Heat Flux Distributions at the Inner Endwall With No Cooling and Nominal
Cooling (Case A) ............................................................................................................................... 122
Figure 6.9 Stanton Number Distributions at the Inner Endwall With No Cooling and
Nominal Cooling (Case A) ................................................................................................................... 123
Figure 6.10 Inlet Temperature Profiles With Nominal Cooling For Case A and Case B ............... 125
Figure 6.11 Heat Flux and Stanton Number Distributions on the Vane Airfoil Surface
With Nominal Cooling – Case B ........................................................................................................... 126
Figure 6.12 Heat Flux Distributions on the Vane Airfoil Surface With Nominal Cooling For Case A and Case B................................................................. 127
Figure 6.13 Stanton Number Distributions on the Vane Airfoil Surface With Nominal Cooling For Case A and Case B.............................................. 128
Figure 6.14 Heat Flux and Stanton Number Distributions at the Inner Endwall With Nominal Cooling – Case B................................................................. 129
Figure 6.15 Heat Flux Distributions at the Inner Endwall With Nominal Cooling For Case A and Case B................................................................. 130
Figure 6.16 Stanton Number Distributions at the Inner Endwall With Nominal Cooling For Case A and Case B................................................................. 131
Figure 6.17 Heat Flux and Stanton Number Distributions at the Inner Endwall For Uniform Runs................................................................. 132
Figure 6.18 Inlet Temperature Profiles For Cooling Levels........................................... 134
Figure 6.19 Heat Flux and Stanton Number Distributions on the Vane Airfoil Surface For Different Cooling Levels With Radial Profile................................. 136
Figure 6.20 Heat Flux and Stanton Number Distributions at the Inner Endwall For Different Cooling Levels With Radial Profile................................. 137
Figure 6.21 Inlet Temperature Profiles for Un-Cooled Uniform Runs of Build 1 and Build 2................................................................................... 139
Figure 6.22 Heat Flux and Stanton Number Distributions on the Vane Airfoil Surface for Un-Cooled Uniform Runs of Build 1 and Build 2.................. 140
Figure 6.23 Pressure Ratio Distributions on the Vane Airfoil Surface for Un-Cooled Uniform Runs of Build 1 (reproduced from data in [114])............... 142
Figure 6.24 Heat Flux and Stanton Number Distributions on Combined Spans for Un-Cooled Uniform Runs of Build 1 and Build 2 .......................................................... 143

Figure 6.25 Inlet Temperature Profiles for Uniform Runs of Build 1 and Build 2........ 144

Figure 6.26 Heat Flux and Stanton Number Distributions on Combined Spans for Uniform Runs of Build 1 and Build 2 .......................................................... 145

Figure 6.27 Heat Flux and Stanton Number Distributions at the Inner Endwall for Un-Cooled Uniform Runs of Build 1 and Build 2 .......................................................... 146

Figure 6.28 Heat Flux and Stanton Number Distributions at the Inner Endwall for Un-Cooled Uniform Runs of Build 1 and Build 2 as Contour Plots (Build 1 – Left, Build 2 – Right) ........................................................................................................ 147

Figure 6.29 Inlet Temperature Profiles for Uniform and Radial Runs With Cooling and No Cooling ........................................................................................................ 149

Figure 6.30 Heat Flux and Stanton Number Distributions on Combined Spans for Uniform and Radial Runs With Cooling and No Cooling ........................................ 150

Figure 6.31 Inlet Temperature Profiles for Uniform Runs of Build 1 and Build 2 With Cooling and No Cooling ..................................................................................... 151

Figure 6.32 Heat Flux and Stanton Number Distributions on Combined Spans for Uniform Runs of Build 1 and Build 2 With Cooling and No Cooling ....................... 151

Figure 6.33 Variations Between Profiles on the Vane Airfoil Surface For Cooled Case155

Figure 6.34 Variations Between Spans on the Vane Airfoil Surface For Cooled Case.. 156

Figure 6.35 Variations Between Profiles (Left) and Between Spans (Right) on the Vane Airfoil Surface For Cooled Case ................................................................. 157
Figure 6.36 Variations Between Profiles on the Vane Airfoil Surface For Un-Cooled Case ........................................................................................................................................................................................................... 160

Figure 6.37 Variations Between Spans on the Vane Airfoil Surface For Un-Cooled Case ........................................................................................................................................................................................................... 161

Figure 6.38 Variations Between Profiles (Left) and Between Spans (Right) on the Vane Airfoil Surface For Un-Cooled Case ........................................................................................................................................................................................................... 162

Figure 6.39 Variations Between Profiles on the Vane Airfoil Surface For Cooled (Left) and Un-Cooled (Right) Cases ........................................................................................................................................................................................................... 164

Figure 6.40 Variations Between Spans on the Vane Airfoil Surface For Cooled (Left) and Un-Cooled (Right) Cases ........................................................................................................................................................................................................... 165

Figure 6.41 Variations Between Profiles on the Vane Airfoil Surface For Cooled and Un-Cooled Cases – Percent Deviations ........................................................................................................................................................................................................... 166

Figure 6.42 Variations Between Spans on the Vane Airfoil Surface For Cooled and Un-Cooled Cases – Percent Deviations ........................................................................................................................................................................................................... 167

Figure 6.43 Endwall Gauge Locations at the Inlet, Passage, and Exit (Not to Scale) .... 170

Figure 6.44 Variations Between Profiles at the Endwall Inlet For Cooled Case ......... 173

Figure 6.45 Variations Between Profiles at the Endwall Passage For Cooled Case...... 174

Figure 6.46 Variations Between Profiles at the Endwall Exit For Cooled Case......... 175

Figure 6.47 Variations Between Profiles at the Endwall Inlet For Un-Cooled Case...... 177

Figure 6.48 Variations Between Profiles at the Endwall Passage For Un-Cooled Case 178

Figure 6.49 Variations Between Profiles at the Endwall Exit For Un-Cooled Case ...... 179

Figure 6.50 Variations Between Profiles at the Endwall Inlet For Cooled (Top) and Un-Cooled (Bottom) Cases ........................................................................................................................................................................................................... 181
Figure 6.51 Variations Between Profiles at the Endwall Passage For Cooled (Left) and Un-Cooled (Right) Cases ................................................................. 182

Figure 6.52 Variations Between Profiles at the Endwall Exit For Cooled (Top) and Un-Cooled (Bottom) Cases ...................................................................... 183

Figure 6.53 Heat Flux Contours at the Endwall For Cooled (Top) and Un-Cooled (Bottom) Cases ...................................................................................... 185

Figure 6.54 Stanton Number Contours at the Endwall For Cooled (Top) and Un-Cooled (Bottom) Cases ...................................................................................... 186

Figure 6.55 Hot Streak Heater Sector and Rake Temperature Measurements .......... 188

Figure 6.56 Inlet Temperature Profiles For Hot Streaks .............................................. 189

Figure 6.57 Effect of Hot Streak Alignment Types With High Magnitude at Mid-Span Suction Surface ..................................................................................... 191

Figure 6.58 Effect of Hot Streak Alignment Types With Medium Magnitude at Mid-Span Suction Surface ..................................................................................... 191

Figure 6.59 Effect of Hot Streak Alignment Types With Medium Magnitude at the Inner Endwall ........................................................................................................... 194

Figure 6.60 Effect of Hot Streak Alignment Types With High Magnitude at the Inner Endwall ........................................................................................................... 196

Figure 6.61 Variations Between Cooling Levels on the Vane Airfoil Surface .............. 200

Figure 6.62 Variations Between Spans on the Vane Airfoil Surface .......................... 201

Figure 6.63 Variations Between Cooling Levels (Left) and Between Spans (Right) on the Vane Airfoil Surface ............................................................................ 202
Figure 6.64 Variations Between Cooling Levels (Left) and Between Spans (Right) on the Vane Airfoil Surface – Percent Deviations ................................................................. 205

Figure 6.65 Variations Between Cooling Levels With Respect to Average of Cooling Levels on the Vane Airfoil Surface ........................................................................ 206

Figure 6.66 Variations Between Cooling Levels With Respect to Average of Cooling Levels (Left) and With Respect to Nominal Cooling (Right) on the Vane Airfoil Surface – Percent Deviations ................................................................. 207

Figure 6.67 Variations Between Cooling Levels With Respect to Average of Cooling Levels (Left) and Between Spans (Right) on the Vane Airfoil Surface – Percent Deviations ................................................................................................................ 208

Figure 6.68 Stanton Number and Net Stanton Reduction at Mid-Span Suction Surface 210

Figure 6.69 Net Stanton Reduction on the Vane Airfoil Surface ........................................... 213

Figure 6.70 Variations Between Cooling Levels at the Endwall Inlet ...................................... 216

Figure 6.71 Variations Between Cooling Levels at the Endwall Passage ............................... 217

Figure 6.72 Variations Between Cooling Levels at the Endwall Exit ..................................... 218

Figure 6.73 Heat Flux Contours at the Endwall For Different Cooling Levels With Radial Profile ........................................................................................................... 220

Figure 6.74 Stanton Number Contours at the Endwall For Different Cooling Levels With Radial Profile ........................................................................................................... 221

Figure 6.75 Net Stanton Reduction Contours at the Inner Endwall For Cooling Levels 223

Figure 6.76 Inlet Temperature Profiles for Purge and Vane Outer Cooling ............................. 227

Figure 6.77 Heat Flux and Stanton Number Distributions on the Vane Airfoil Surface With Purge Flow Variation ......................................................................................... 229
Figure 6.78 Heat Flux and Stanton Number Distributions at the Inner Endwall With
Purge Flow Variation ................................................................. 230

Figure 6.79 Heat Flux and Stanton Number Distributions on the Vane Airfoil Surface
With and Without Vane Outer Flow ............................................... 233

Figure 6.80 Heat Flux and Stanton Number Distributions at the Inner Endwall With
and Without Vane Outer Flow ....................................................... 234

Figure 6.81 Effect of Vane Outer Cooling Compared to Other Cooling Levels on the
Vane Airfoil Surface ........................................................................ 237

Figure 6.82 Effect of Vane Outer Cooling Compared to Other Cooling Levels at the
Inner Endwall – Deviations .............................................................. 238

Figure 6.83 Effect of Vane Outer Cooling Compared to Other Cooling Levels at the
Inner Endwall – Percent Deviations ................................................... 239

Figure 6.84 Comparison of Deviations Caused by Cooling Levels (With Radial
Profile) and Profiles (at Nominal Cooling) on the Vane Airfoil Surface......... 242

Figure 6.85 Comparison of Percent Deviations Caused by Cooling Levels (With
Radial Profile) and Profiles (at Nominal Cooling) on the Vane Airfoil Surface .... 243

Figure 6.86 Comparison of Span-Wise Deviations Caused by Cooling Levels (With
Radial Profile) and Profiles (at Nominal Cooling) on the Vane Airfoil Surface .... 244

Figure 6.87 Comparison of Span-Wise Percent Deviations Caused by Cooling Levels
(With Radial Profile) and Profiles (at Nominal Cooling) on the Vane Airfoil
Surface................................................................................................. 245

Figure 6.88 Comparison of Deviations in Heat Flux Caused by Cooling Levels (With
Radial Profile) and Profiles (at Nominal Cooling) at the Inner Endwall ............. 246
Figure 6.89 Comparison of Percent Deviations in Heat Flux Caused by Cooling Levels
(With Radial Profile) and Profiles (at Nominal Cooling) at the Inner Endwall...... 247
Figure 6.90 Comparison of Deviations in Stanton Numbers Caused by Cooling Levels
(With Radial Profile) and Profiles (at Nominal Cooling) at the Inner Endwall...... 248
Figure 6.91 Comparison of Percent Deviations in Stanton Numbers Caused by Cooling
Levels (With Radial Profile) and Profiles (at Nominal Cooling) at the Inner
Endwall.................................................................................................................... 249
Figure A.1 Stanton Number Distributions Using Local Temperature (Left) and
Average Temperature (Right) ................................................................................. 274
Figure A.2 Effect of Using Local Temperature in Stanton Numbers for Radial Profile 275
Figure A.3 Effect of Using Local Temperature in Stanton Numbers for Uniform
Profile ..................................................................................................................... 276
Figure A.4 Effect of Using Local Temperature in Stanton Numbers ......................... 277
Figure A.5 Stanton Number Distributions Using Local Temperature (Left) and
Average Temperature (Right) ................................................................................. 280
Figure A.6 Effect of Coolant Mass in Stanton Numbers ............................................. 281
Figure A.7 The Inlet Temperature Profile for the Un-Cooled Radial Profile ............... 282
Figure A.8 The Influence of Extended Inlet Temperature Profile on 5% Span Stanton
Numbers .................................................................................................................. 283
Figure A.9 The Influence of Extended Inlet Temperature Profile on Inner Endwall
Stanton Numbers .................................................................................................... 284
# LIST OF TABLES

Table 4.1 Cooling Holes in the Vane Section ........................................................................... 45

Table 4.2 Gauge Locations Used For Measurements .......................................................... 48

Table 5.1 Experimental Matrix ............................................................................................ 73

Table 5.2 Typical Experimental Conditions for Which Data Were Obtained ............... 75

Table 5.3 All Runs Presented For Temperature Profiles and Cooling Levels ............... 79

Table 5.4 Repeatability of Experimental Input Conditions For Different Groups ......... 89

Table 6.1 Experimental Parameters for Runs Presented For Temperature Profile and Cooling Level Effects ....................................................................................................................... 109

Table 6.2 Runs Presented For Purge and Vane Outer Coolant Effects ......................... 226
**NOMENCLATURE AND ABBREVIATIONS**

*Description*

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$a_1$</td>
<td>Scale</td>
<td>(K/ohm)</td>
</tr>
<tr>
<td>A</td>
<td>Area</td>
<td>($m^2$)</td>
</tr>
<tr>
<td>AFTRF</td>
<td>Axial Flow Turbine Research Facility</td>
<td></td>
</tr>
<tr>
<td>BL</td>
<td>Boundary Layer</td>
<td></td>
</tr>
<tr>
<td>BR</td>
<td>Blowing Ratio</td>
<td></td>
</tr>
<tr>
<td>CA</td>
<td>Compound Angle hole</td>
<td></td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
<td></td>
</tr>
<tr>
<td>CVP</td>
<td>Counter-rotating Vortex Pair</td>
<td></td>
</tr>
<tr>
<td>$C_D$</td>
<td>Discharge coefficient</td>
<td></td>
</tr>
<tr>
<td>$C_p$</td>
<td>Specific heat</td>
<td>(J/kg.K)</td>
</tr>
<tr>
<td>$d$</td>
<td>Substrate thickness</td>
<td>(in)</td>
</tr>
<tr>
<td>D</td>
<td>Diameter</td>
<td>(m or ft)</td>
</tr>
<tr>
<td>DAS</td>
<td>Data Acquisition System</td>
<td></td>
</tr>
<tr>
<td>DR</td>
<td>Density Ratio</td>
<td></td>
</tr>
<tr>
<td>ETHZ</td>
<td>Swiss Federal Institute of Technology</td>
<td></td>
</tr>
<tr>
<td>FAV</td>
<td>Fast Acting Valve</td>
<td></td>
</tr>
<tr>
<td>FF</td>
<td>Flow Function</td>
<td></td>
</tr>
<tr>
<td>FFT</td>
<td>Fast Fourier Transform</td>
<td></td>
</tr>
<tr>
<td>G</td>
<td>Pre-amp gain</td>
<td></td>
</tr>
<tr>
<td>GEA</td>
<td>General Electric Aviation</td>
<td></td>
</tr>
<tr>
<td>GTL</td>
<td>Gas Turbine Laboratory</td>
<td></td>
</tr>
<tr>
<td>$h$</td>
<td>Heat transfer coefficient</td>
<td>(W/m$^2$.K)</td>
</tr>
<tr>
<td>I</td>
<td>Momentum flux ratio or current</td>
<td>(A for current)</td>
</tr>
<tr>
<td>Abbreviation</td>
<td>Definition</td>
<td>Unit</td>
</tr>
<tr>
<td>--------------</td>
<td>---------------------------------------------------------------------------</td>
<td>---------------</td>
</tr>
<tr>
<td>ILPT</td>
<td>Isentropic Light Piston Facility</td>
<td></td>
</tr>
<tr>
<td>k</td>
<td>Thermal conductivity of substrate or fluid</td>
<td>(W/m.K)</td>
</tr>
<tr>
<td>L</td>
<td>Length of cooling hole</td>
<td>(m or in)</td>
</tr>
<tr>
<td>LCF</td>
<td>Large Cooling Facility</td>
<td></td>
</tr>
<tr>
<td>LE</td>
<td>Leading Edge of the airfoil</td>
<td></td>
</tr>
<tr>
<td>M</td>
<td>Mach Number</td>
<td></td>
</tr>
<tr>
<td>MIT</td>
<td>Massachusetts Institute of Technology</td>
<td></td>
</tr>
<tr>
<td>MP</td>
<td>Mid-Passage</td>
<td></td>
</tr>
<tr>
<td>N&lt;sub&gt;corr&lt;/sub&gt;</td>
<td>Corrected Speed</td>
<td>(rpm/√K)</td>
</tr>
<tr>
<td>NGV</td>
<td>Nozzle Guide Vane</td>
<td></td>
</tr>
<tr>
<td>NHFR</td>
<td>Net Heat Flux Reduction</td>
<td></td>
</tr>
<tr>
<td>NSR</td>
<td>Net Stanton Reduction</td>
<td></td>
</tr>
<tr>
<td>Nu</td>
<td>Nusselt Number</td>
<td></td>
</tr>
<tr>
<td>OSU</td>
<td>The Ohio State University</td>
<td></td>
</tr>
<tr>
<td>P</td>
<td>Pressure</td>
<td>(N/m&lt;sup&gt;2&lt;/sup&gt; or psi)</td>
</tr>
<tr>
<td>PR</td>
<td>Pressure Ratio</td>
<td></td>
</tr>
<tr>
<td>PS</td>
<td>Pressure Surface</td>
<td></td>
</tr>
<tr>
<td>q′</td>
<td>Heat flux</td>
<td>(W/m&lt;sup&gt;2&lt;/sup&gt;)</td>
</tr>
<tr>
<td>R</td>
<td>Specific gas constant or resistance</td>
<td>(kJ/kg.K) or (ohms)</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number</td>
<td></td>
</tr>
<tr>
<td>Ro</td>
<td>Rotation Number</td>
<td></td>
</tr>
<tr>
<td>RTD</td>
<td>Resistance Temperature Device</td>
<td></td>
</tr>
<tr>
<td>SS</td>
<td>Suction Surface</td>
<td></td>
</tr>
<tr>
<td>St</td>
<td>Stanton Number</td>
<td></td>
</tr>
<tr>
<td>t</td>
<td>Time</td>
<td>(sec)</td>
</tr>
<tr>
<td>T</td>
<td>Temperature</td>
<td>(K)</td>
</tr>
<tr>
<td>T&lt;sub&gt;lab&lt;/sub&gt;</td>
<td>Lab temperature</td>
<td>(K)</td>
</tr>
<tr>
<td>TE</td>
<td>Trailing Edge of the airfoil</td>
<td></td>
</tr>
<tr>
<td>TRF</td>
<td>Turbine Research Facility</td>
<td></td>
</tr>
<tr>
<td>TTF</td>
<td>Turbine Test Facility</td>
<td></td>
</tr>
</tbody>
</table>
Symbols

\( \rho \) Density of substrate or flow \( (\text{kg/m}^3) \)
\( \eta \) Film effectiveness
\( \gamma \) Ratio of specific heats, \( C_p/C_v \) at constant pressure and volume, respectively
\( \Delta q_r \) Net heat flux reduction
\( \alpha \) The streamwise injection angle
\( \beta \) The lateral injection angle
\( \sigma \) Stefan-Boltzmann constant \( (\text{W/m}^2\cdot\text{K}^4) \)
\( \varepsilon \) Emissivity of the surface
\( \dot{m} \) Mass flow rate \( (\text{kg/sec}) \)
\( \Delta \) Change

Subscripts

avg Average
ave Area-weighted average
aw Adiabatic wall
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>c</td>
<td>Coolant</td>
</tr>
<tr>
<td>f</td>
<td>With film cooling</td>
</tr>
<tr>
<td>l</td>
<td>Lower Kapton gauge</td>
</tr>
<tr>
<td>phys</td>
<td>Physical</td>
</tr>
<tr>
<td>ref</td>
<td>Reference</td>
</tr>
<tr>
<td>surr</td>
<td>Surroundings</td>
</tr>
<tr>
<td>u</td>
<td>Upper Kapton gauge</td>
</tr>
<tr>
<td>w</td>
<td>Wall</td>
</tr>
<tr>
<td>0</td>
<td>Without film cooling or total values of temperature or pressure</td>
</tr>
<tr>
<td>∞</td>
<td>Mainstream gas</td>
</tr>
</tbody>
</table>
CHAPTER 1

INTRODUCTION

The design of air-cooled gas turbines is driven by many competing factors, such as efficiency, power, emissions, and component life. Modern design trends continue to move towards ever increasing turbine inlet temperatures as designers strive to extract more power output and higher efficiencies. Even small gains in overall turbine efficiency can be the determining factor in commercially successful designs, as energy costs and environmental concerns add additional pressures to turbine designers.

Under these circumstances, maintaining high turbine inlet temperatures is critical. This task requires sufficient amount of cooling on all turbine components in order to avoid a possible failure due to burnout. The practice in the engine design community has been to bleed a nontrivial amount of air from the compressor and to use it for hot section cooling purposes. As bleeding significant cooling air from the compressor has the penalty of performance losses associated with it, efforts spent in refinement of film cooling technology are definitely crucial for success in utilization of the compressor air more effectively.

Film cooling has been one of the widespread cooling techniques for airfoil design. It is a technique that aims to protect the metal surface from the turbine hot gas environment by using the coolant as a protective thermal shield through the boundary layer. This coolant is supplied through cooling holes located on the surface, which are machined into the internal blade cooling passages. As a result of this process, the supplied coolant insulates the metal surface from the hot gas path, and the detrimental effect of hot free stream gas on the metal surface is reduced. If done correctly, this process increases the life expectancy of the turbine component significantly. Consequently, film cooling has been one of the major approaches to thermal protection of metal surfaces.
In the absence of experimental results relevant to film cooling designs, it would be difficult to design cooling schemes in a cost effective manner for gas turbine engines. However, because the number of experiments that can be performed is limited by time and cost issues, an engine designer relies on computational fluid dynamics (CFD) codes that have incorporated into them modeling of the cooling scheme that is based on the available data base. More specifically for gas turbine cooling applications, the compromise between the engine performance and the coolant amount is essentially an optimization study that requires the support of computational tools, which would otherwise not be practical to handle via experimentation. In order to minimize the necessary coolant amount for protection of turbine components during engine operations, the designer performs predictions for the film cooling effectiveness and component heat loads, and needs to accomplish this with some degree of certainty making the predictive capability of these codes an important factor that goes through the design process. The cooling models used in these codes are engine-company specific and are generally based on measurements taken using flat plate or cascade geometries. Until this point in time, a data set resembling the one reported in this thesis has not been available to the industry. Confidence in a particular approach requires a data set of high quality for which the flow physics is properly duplicated.

Obviously, the most realistic data would be that obtained from an engine operating under design conditions. Unfortunately, the harsh environment associated with the engine makes this an impossible situation. As a result, an alternate method has been selected that involves using actual engine hardware operating at design corrected conditions, which represents the operating engine environment as closely as one can under controlled laboratory conditions, while enabling control of specific variables of interest, such as vane inlet temperature profiles and cooling gas flow rate.

1.1 Significance of the Problem

This research focuses on film cooling, and its effects on the flow physics of the vane component of a rotating turbine stage operating at design corrected conditions. A quick glimpse at the literature presents an overwhelming amount of research that has been done related to film cooling. However, most of this research has been performed
using very simple flow geometries, such as flat plates or a bit more realistically cascades that give some insight into the effects of film cooling, but overall, the associated flow physics is far from simulating a real engine environment.

Operating full-stage rotating turbines at design corrected conditions is the most sophisticated experimental approach; however, successfully doing so for a fully-cooled machine has been a long time in development. The reason behind this is not that the researchers do not see its importance, but rather it is not an easy task due to the operation complexities, instrumentation difficulties, and accuracy issues. For example, in order to be able to capture unsteady effects such as wake-blade interaction, fast response instrumentation and high-speed data acquisition systems are required, whose development dates back to the hypersonic research era. It is not only the instrumentation quality or the knowledge of bringing these necessities together that are essential, but more importantly understanding how all these pieces go together and interpreting the data correctly are what make this type of experimentation a possibility.

Despite the extensive research in this area, until very recently, detailed film-cooling measurements for a fully-cooled turbine stage did not exist in the open literature. The recent study of Haldeman et al. [1-2] reports fully-cooled turbine stage measurements, but vane measurements for those experiments were limited to the mid-span. Hence, detailed film-cooled vane data still does not exist in the open literature. This missing data set is very critical to the gas turbine industry for improving the capabilities of the current state-of-the-art design of these machines so that more efficient cooled designs with extended turbine life can be accomplished. The research presented in this thesis is intended not only to create an empirical study on the effects of different types of variables on the vane heat transfer, but also to provide the designer with high quality data.

1.2 Scope and Objectives of the Research

As part of the Heat Transfer initiative of the University Strategic Alliance (USA) Program sponsored by General Electric Aviation (GEA), a full-scale stage and one-half high-pressure turbine has been designed and manufactured by GEA, and instrumented, and operated at the OSU Gas Turbine Laboratory (GTL). The research program has been
performed in stages beginning with measurement of the aerodynamics and heat transfer (and subsequent CFD predictions) of an un-cooled version of the turbine, followed by a detailed clocking study to investigate the influence of vane clocking on aeroperformance (all referred to as Build 1), and concluding with the present measurement program involving replacement of the high-pressure turbine un-cooled vane row with a fully-cooled vane row and the addition of disk cavity purge flow, while retaining the un-cooled rotor stage (known as Build 2, Entry 1). The measurements were performed using the OSU GTL Turbine Test Facility (TTF) that has been used many times for previous rotating turbine research programs. The GTL has pioneered the development of fast response instrumentation and has invested heavily in high-speed data acquisition systems, thus making it possible to obtain measurement of both time-averaged and time-accurate data for a very large number of individual sensors.

This thesis is focused on the final phase of the USA heat-transfer program (Build 2, Entry 1) noted above and specifically on the cooling gas injection of the cooled high-pressure turbine vane. The cooled vane row contains cooling holes on both the endwalls and the airfoil. The addition of a new vane row made it necessary to completely re-instrument the vane row. In addition, Build 2 utilizes a combustor emulator (not included in Build 1), that is located upstream of the vane row, making it possible to produce upon demand different temperature profiles at the vane inlet. The Build 2 data matrix also provides information for different coolant mass flows, thus helping to establish an understanding of the effect of these parameters on heat transfer and cooling effectiveness on different regions of the vane. Comparing the results of the cooled cases with those of the un-cooled cases performed in both Build 1 and Build 2 will emphasize the importance of film cooling for the gas turbine design community. The difficulty with comparison of the un-cooled case for selected regions of the vane for the Build 2 geometry will be described in detail. The free stream turbulence effects have not been investigated to this point since locating a turbulence grid upstream of the vane attenuates the vane inlet temperature profiles. As a result of the construction technique used in construction of the combustor emulator, the free stream turbulence at the inlet to the vane row is very low. For the purposes of completeness, the plan is to use the turbulence grid just mentioned
during a later entry so that the influence of free stream turbulence on the film-cooled vane and blade of a cooled turbine can be investigated.

Detailed film-cooled vane data for a realistic cooling-hole configuration with a downstream rotor still do not exist in the open literature. The goal of the current study is to fill this void by providing an extensive set of heat-transfer data for a fully-cooled vane exposed to different inlet temperature profiles and for a range of blowing ratios, while matching design corrected conditions. This data set aims to provide the information necessary to refine the understanding of film cooling on vane heat transfer so that the design philosophy can make significant positive strides. The database is intended to give information about not only the interaction of film coolant flow with the hot gas path flow, but also some information regarding the unsteady and rotational effects of the downstream blade row on the vane surface heat transfer.
CHAPTER 2

FILM COOLING FUNDAMENTALS

The flow field associated with the high-pressure turbine stage is complex and three-dimensional in nature. There are many factors contributing to this complexity, the major ones being boundary layer (BL) transition, wake-blade interactions, secondary flows such as vortices created by the endwall, tip leakage flows, shock waves due to high speed, rotational effects, surface curvature effects, and free stream turbulence. One can find an enormous number of investigations in the literature that focus on the effect of one or a combination of all these flow phenomena on engine components. Film cooling is another complexity of the flow field.

![Figure 2.1 The Three-Dimensional Separation of a Boundary Layer Entering a Turbine Cascade [3]](image)

There are many proposed models of passage flow demonstrating the interactions with the mainstream flow through an airfoil passage. One of the early models proposed by Langston [3] describes the vane endwall flow pattern, which is highly dominated by
vortical structures, as shown in Figure 2.1. A horseshoe vortex is defined as a leading edge vortex that is formed by the separation of the inlet boundary layer at the cascade endwall. Upon stagnation, the flow undergoes a 3-D separation into two branches at the leading edge, one along the suction surface and the other along the pressure surface. This location is at the intersection of the airfoil with the endwall. One leg of this vortex merges with the passage vortex, while the other leg (labeled as counter vortex in Figure 2.1) remains in the suction surface endwall corner. The passage vortex is a combination of the crossflow within the cascade passage, the horseshoe leg that is originally formed on the pressure surface, and the portion of the mainstream flow that is entrained in this passage flow. The counter vortex rotates in the opposite direction to the passage vortex, and is much smaller in magnitude. Langston et al. [4] showed that this three-dimensional secondary flow is responsible for a decrease in the pressure differences between the pressure and suction surfaces, resulting in a consequent growth in aerodynamic loss through the passage.

Blair [5] suggested that there are three main mechanisms producing secondary flows. The mainstream flow is turned by the airfoil (of blade or vane) as it passes through the passage creating a cross channel flow by the endwall from the pressure surface of an airfoil towards the suction surface of a neighboring airfoil, and rolls up into a passage vortex, as in the case of a vane endwall. The second mechanism is the relative motion between the blade tip and the endwall. As the endwall moves toward the suction surface, at sufficiently high rotational speeds, a vortex is generated along the suction surface. A third mechanism is the tip leakage flow, which is caused by the clearance gap of the blade, and creates a leakage vortex over the tip. These last two mechanisms are specific to the blade and are not observed on the vane component due to it being stationary.

Other than the leakage flow, all other vortices are attributed to the interaction with the endwall. Although in above paragraph the tip leakage flow is defined as a secondary flow, in the literature the secondary flows are often confined to those flows created by the endwall only. However, regardless of the definition, the interaction among all of these flows may be very strong, which can result in significant performance losses. Denton [6] suggests that loss is a combination of many factors, and the effects of profile loss
(viscosity and separation effects in boundary layer), leakage loss, and endwall loss cannot be distinguished easily. As a result, all these losses can be counted as secondary flows, which contribute to the overall performance penalty of the engine.

There are many parameters of interest regarding film-cooling applications. For example, a designer needs to know the location and the magnitude of the heat load occurring on the airfoil surface due to the hot gas path, in order to improve cooling techniques. Without film cooling, the convective heat flux, \( q_0'' \), to the surface is defined by:

\[
q_0'' = h_0 (T_\infty - T_w)
\]  

(1)

where \( h_0 \) is the heat-transfer coefficient without film cooling flow, and \( T_\infty \) and \( T_w \) are the mainstream temperature and the airfoil wall temperature, respectively. Instead of \( T_\infty \), sometimes a local recovery temperature definition is used since the external gas temperature is not constant everywhere. With the addition of film cooling, the wall is exposed to the coolant film layer and not the mainstream gas, and the heat transfer to the surface becomes:

\[
q_f'' = h_f (T_{aw} - T_w)
\]  

(2)

where \( h_f \) is the heat-transfer coefficient and \( q_f'' \) is the heat flux with the effect of film cooling. \( T_{aw} \) is defined as the adiabatic wall temperature at which there is no heat transfer to the wall. The above formulation assumes that \( T_{aw} \) is the film temperature, which is the mixture of the mainstream gas and coolant temperatures, and is assumed to be the effective temperature acting on the wall. Therefore, the adiabatic wall temperature is somewhere between the coolant and the mainstream temperatures. The assumption that the adiabatic wall temperature is the effective mainstream temperature is valid as long as the thermal boundary layer developing over the surface stays thin compared to coolant jet thickness [7].
A parameter that becomes important when discussing film cooling is film effectiveness, \( \eta \), which is a measure of how effective the cooling is in keeping the surface temperature cool, and is referred to as the dimensionless adiabatic wall temperature.

\[
\eta = \frac{(T_\infty - T_{aw})}{(T_\infty - T_c)} \tag{3}
\]

where \( T_c \) is the coolant temperature. Although the coolant itself reduces heat transfer to the airfoil by reducing the gas temperature in contact with the surface, the injection process can result in an increase in heat transfer coefficient because of the disturbance it creates in the flow. Hence, both an increase and a decrease in heat transfer occur simultaneously, which makes a definition of the net film cooling effect on the surface necessary. Therefore, another parameter of interest is the Net Heat Flux Reduction, which is a measure of overall film cooling performance, and is defined by Sen et al. \[8\] as:

\[
NHFR = \Delta q_r = \frac{q_0^{\infty} - q_r^{\infty}}{q_0^{\infty}} = 1 - \frac{q_r^{\infty}}{q_0^{\infty}} = 1 - \frac{h_f (T_{aw} - T_w)}{h_0 (T_\infty - T_w)} \tag{4}
\]

Considering the possibility of having negative heat flux in the flow field, this definition was slightly modified later by Mathison et al. \[9\] as:

\[
NHFR = \Delta q_r = -\frac{q_0^{\infty} - q_r^{\infty}}{|q_0^{\infty}|} \tag{5}
\]

With equation (5), a positive NHFR suggests that the coolant reduces heat transfer, while a negative value indicates an increase in heat transfer due to the coolant addition. Overall, the heat-transfer coefficient and film effectiveness characterize the quality of film cooling. It is obvious that for effective cooling on the surface, \( h \) needs to be minimized while \( \eta \) needs to be maximized as much as possible.

Generally, the two key flow parameters of interest are the blowing ratio (BR) and the temperature ratio of the coolant to mainstream, \( T_c/T_\infty \). Blowing ratio can be related to
the pressure ratio, whereas the temperature ratio is related to the density ratio, DR, of coolant to mainstream. For this reason density ratio has been one of the parameters investigated in many flat plate studies. Both parameters are important in quantifying film-cooling effectiveness. BR is referred to as mass flux ratio, and the two are often used interchangeably in the literature.

\[
BR = \frac{\rho_c U_c}{\rho_\infty U_\infty} = \frac{P_c M_c (\gamma R_c T_c)^{1/2}}{P_\infty M_\infty (\gamma R_c T_c)^{1/2}} \tag{6}
\]

\[
DR = \frac{\rho_c}{\rho_\infty} = \frac{P_c R_c T_c}{P_\infty R_c T_c} \tag{7}
\]

Here, \(\rho_c\) and \(\rho_\infty\) are the coolant and mainstream densities, and \(U_c\) and \(U_\infty\) are their corresponding velocities.

The coolant jet trajectory can be scaled by three different parameters, one of which is discussed above (BR), and this trajectory is connected to the jet separation. BR is a good measure of the jet trajectory as long as the coolant jet stays attached to the surface. If the blowing ratio is beyond an optimum level, the jet can penetrate into the mainstream. This phenomenon is called lift-off (or blow-off) and is responsible for significant decreases in effectiveness, and in this case a better scale for film effectiveness is needed. Another parameter is the momentum flux ratio, I, which is the combination of blowing ratio and density ratio. For lift-off cases, the jet trajectory can be better quantified by this parameter rather than the blowing ratio, as long as there is no gross separation immediately after the ejection location.

\[
I = \frac{\rho_c U_c^2}{\rho_\infty U_\infty^2} = \frac{BR^2}{DR} \tag{8}
\]

A third parameter used for jet trajectory definition is the velocity ratio, Vr. It is usually used for the cases where the coolant jet directly penetrates into the mainstream. Bernsdorf
et al. [10] experimentally observed that the blowing rate of the jet correlates well with the momentum flux ratio, I. Another observation from their study was that for a fixed BR, the blowing of the jet decreases with an increase in the coolant jet density. Thole et al. [11] performed another sample study measuring cooling jet separation for different separation cases using all three parameters. They also concluded that the jet separation was best scaled with momentum flux ratio, I.

Usually, the heat transfer to or from the surface is presented in terms of the non-dimensional Stanton or Nusselt Number in order to normalize the data:

\[
St = \frac{h}{\rho_c u_c c_p} = \frac{q''}{m \frac{\rho \overline{C_p}}{A_{ref}} [(C_p T)_{ref} - (C_p T)_w]} \tag{9}
\]

\[
Nt_k = \frac{h x}{k} = \frac{q''}{k[T_{ref} - T_w]} \tag{10}
\]

When Stanton Number is used in the turbine environment, which is also the parameter used in this thesis, \(A_{ref}\) is used as the reference area that is usually taken as the high-pressure vane inlet area, and \((C_p T)_{ref}\) refers to the specific heat evaluated at the total temperature measured by the vane inlet rakes. However, one should be careful in using this definition for a film cooling experiment, since the effective vane throat area changes depending on the cooling amount. This means that if the reference area is assigned as the vane throat area, the effect of a change in area caused by the differences between different cooling levels will also be reflected in Stanton Number. The parameter \(T_w\) is defined as the wall temperature. The temperature difference between the mainstream and the wall temperature, \((T_{ref} - T_w)\), is referred to as the driving temperature.

When Stanton Number is used for the presentation of the data instead of heat flux values, a similar parameter to NHFR is often used in order to quantify the overall film cooling effectiveness. Net Stanton Reduction is defined in a similar way:
Here, \( \text{St}_0 \) stands for the Stanton Number without cooling, and \( \text{St}_f \) is for the case with film cooling. A substitute to this definition is defined by equation (12) in order to handle the effect of negative heat flux, as was previously introduced for NHFR:

\[
NSR = 1 - \frac{\text{St}_f}{\text{St}_0}
\]

\[\text{NSR} = \frac{\text{St}_0 - \text{St}_f}{|\text{St}_0|}\] (12)

Another parameter that is investigated for film cooling applications is the discharge coefficient. It is used to size the film cooling holes so that the actual BR and I ratios can be determined, and is defined as the ratio of actual to ideal mass flow rates through the cooling hole:

\[
C_D = \frac{\dot{m}_{\text{actual}}}{\dot{m}_{\text{ideal}}}
\]

Discharge coefficient is usually determined via empirical correlations. This coefficient is highly dependant on the length-to-diameter ratio (L/D) of the film hole, the injection angle, as well as the inlet/exit flow conditions, and the geometry. Once the pressure difference through the cooling hole is measured, the ideal mass flow rate can be calculated using isentropic flow relationships. Using the above relation, one can then calculate the actual mass flow rate. The actual mass flow rate could be obtained at the choked holes directly instead of using a discharge coefficient. With the external pressure and hole cross-sectional area information, the blowing ratio through a particular hole can be calculated. Recall that for a conventional vane, one may have several hundred cooling holes so doing this kind of calculation for individual holes can become quite an effort.

It is important to correctly size the cooling holes in order to obtain the desired cooling performance. Gritsch et al. [12] investigated the discharge coefficients for different shaped holes, and they found that this parameter was strongly dependent on
internal and external flow conditions, as well as the pressure ratio. The discharge coefficients of expanded holes were found to be higher than that of cylindrical holes, which resulted in higher mass flow rates. In a subsequent study [13], they observed that increasing the angle of the hole decreased discharge coefficients because of an increase in losses at the entry to the hole.

So far, the flow parameters that the film cooling performance depends on have been discussed. It is obvious that film cooling must also depend on the geometric parameters, such as the angle and the shape of the holes, as well as their locations with respect to each other.

The length-to-diameter ratio (L/D) should be high enough to let the cooling flow develop so that the hole entry does not influence the flow establishment. One of the early studies by Goldstein et al. regarding shaped holes [14] observed that the cooling ejection was independent of the geometry of the hole if the L/D ratio was maintained at a value in excess of 5.2.

Discrete film cooling holes is the main cooling scheme used on airfoils for cooling purposes. Although there are alternatives such as slots (a form of which is used at the trailing edge of the vane and rotor airfoils of this turbine stage) or even a micro cooling process called transpiration, none of these are as common as discrete cooling. Slot injection is also considered through combustor-to-nozzle gaps or at adjacent nozzle interfaces only, as these are also some of the leakage producing locations in the turbine. Discrete cooling holes cause jet separation much earlier than slots, because there is room around the jet for the mainstream flow to pass through. That is, slot injection is more effective in terms of cooling coverage; however, it would structurally weaken the airfoil if they were located on the airfoil. It has been suggested that transpiration film cooling is the most advantageous cooling techniques of all, but it is not applied on designs yet due to difficulties such as manufacturing, airfoil strength, and cost.

There are two angles defining the orientation of a cooling hole. The streamwise injection angle, α, is the angle relative to the surface in the mainstream (axial) direction. The other angle is the lateral injection angle, β, which is the angle from the mainstream direction. A cooling hole may be oriented purely in the axial or the radial direction. If it is
a combination of the two, then the hole is called a compound angle (CA) hole. As far the shape goes, cylindrical holes are common. However, more recent designs have improved shapes with expanded exits, such as fan-shaped or laid-back. Fan-shaped holes have a lateral expansion section at the exit of the hole, while laid-back shapes have expansion into the surface of the airfoil. A combination of the two is more common in use due to easiness in manufacturing. Schmidt et al. [15] observed that when the round hole had a compound angle, the effectiveness increased in the case of a flat-plate geometry. With the addition of forward expansion to the shape, further improvement was obtained. Gritsch et al. [16] confirmed these results, this time in a supersonic cross flow. Effectiveness was found to be increasing even further compared to subsonic cases. More detailed research on this topic can be found in Bunker’s [17] recent review of shaped film cooling technology.

The distance (pitch) between cooling holes and spacing between the rows also affect cooling coverage. The holes may be distributed through different rows along the airfoil surface in order to increase film effectiveness, so that the limitation of a single row of holes due to lift-off can be avoided. This type of cooling scheme consisting of many rows is referred to as full-coverage film cooling. It results in an effective cooling, but is costly in terms of aerodynamics. Sinha et al. [18] studied the interaction between two rows of holes for different density ratios. The flow field of the second row of holes differed from that of the first in terms of having a thicker boundary layer and a three-dimensional disturbance already created by the first row resulting in greater penetration of the coolant jet into the mainstream. The second row was found to have lower turbulence, which was explained to be due to lower velocity gradients in a thicker boundary layer. The rows may be designed to be staggered in order to increase the strength of the airfoil structure. In another effort, Ligrani and Ramsey [19] examined the effects of different hole configurations and blowing ratios for two staggered rows. They observed a significant coalescence of coolant injected from the upstream row with the downstream row resulting in a span-wise periodic coolant coverage. As blowing ratio increased beyond 1.0, the effectiveness was found to decrease due to lift-off.
CHAPTER 3
LITERATURE REVIEW

The previous chapter introduced a portion of the literature and highlighted the main parameters of interest in the film cooling area. These studies were mostly flat-plate experiments. This chapter presents a review of the film cooling literature from both an experimental and a computational perspective. The work is reviewed from an experimental setup perspective, that is, it will classify the research as flat plate, cascade, or rotating turbine measurements. The rotating turbine measurements are far fewer in number due to the complexity of the setup. Since the current research is performed in a rotating turbine configuration, this last group of studies will be discussed in more detail with the goal of establishing an acquaintance for the reader to the facilities. These experiments will be extended to cover studies both with and without film cooling. The second part of the chapter will give an overview of the computational work done in the film cooling area since the data collected in this research ultimately will be used to improve film-cooling models in conjunction with predictive techniques.

There have been many studies performed in this field focusing on different parts of the turbine vane and blade rows. These studies generally concentrate either on the aerodynamics or on the heat transfer side of the problem. For the computational area, the effort has been to improve predictive techniques with available experimental data sets, but mainly for un-cooled cases. Although the efforts in the two areas seem to be advanced simultaneously, as Dunn [20] suggests, there has always been a time lag between the experimental investigation and the CFD code improvements. The CFD codes are generally based on older experimental results, rather than being regularly updated with the current knowledge from the most recent experimental studies. It is suggested in [20] that having both communities communicate with each other so that all the effort
devoted to this topic can result in more improved and realistic CFD predictions should help to overcome this time lag.

3.1 Experimental Data Base

Film cooling is one of the important subjects in the turbine heat transfer area that was introduced to the turbomachinery community in the 1950’s. Since then, there have been significant improvements in the overall efficiency of gas turbines by allowing for higher firing temperatures and subsequently longer lives of turbine components. Due to the nature of the application, the scope of film cooling research has been quite widespread covering a wide range of topics in both the experimental and computational arena, but with relatively few attacking the flow consistent with a rotating environment.

3.1.1 Flat-Plate Experiments

Flat-plate studies have been popular due to the ease of studying the effects of individual or a combination of parameters. One of the earliest studies on film effectiveness is that of Goldstein et al. [14] published in the 1970’s, who reported studies of film cooling effects due to hole geometry and coolant density. They observed that film effectiveness was improving with shaped holes by causing a reduction in momentum of the coolant as well as a Coanda effect, which caused the jet to hug the surface, resulting in less coolant penetration into mainstream.

Following this study, there have been numerous experiments conducted in the literature using the flat-plate configuration, concentrating on the geometry of holes, such as shape and orientation, on interaction of rows of holes, or on flow parameters such as blowing and density ratios, and turbulence effects. For example, Saumweber and Schulz [21] studied the interaction of two staggered rows with shaped holes, and reported that high blowing ratios can be used more effectively with the downstream row. Also, Bons et al. [22] showed that high free-stream turbulence does not always decrease effectiveness, but rather its effect depends on the location as well as on the blowing ratio. At low blowing ratios, the turbulence was found to be decreasing the effectiveness behind the injection holes, but for high blowing ratios it resulted in higher effectiveness due to causing a reduction in lift-off strength. High turbulence also increased the effectiveness between the holes, significantly.
The research mentioned in the previous chapter was mostly flat-plate experiments, which all contribute to defining the parameters that describe the turbine flow field. The space allowed here would not be enough to mention all the work done using flat plates. At this point, it is tempting to move forward to the more complicated configurations that reveal more information regarding the engine environment.

### 3.1.2 Cascade Data Base

Although film cooling is both an aerodynamics and a heat transfer problem, many of the researchers tended to focus on aerodynamics rather than on heat transfer due to more interest in performance penalties. But one still finds a comprehensive body of configuration-specific experiments in heat transfer in the literature as well, both with and without film cooling, and concentrating on different sections of the vane and the blade. In order to reflect the geometry of the airfoil surfaces of interest better, mainly cascade setups are used. Hence, another big class of research in film cooling is devoted to cascade experiments.

A pioneering work performed by Blair [5] in the 1970s described turbine endwall heat transfer with coolant injection. Coolant was injected onto the endwall through a slot located upstream of the leading edge plane. The film effectiveness was found to be high in the immediate slot region, but there was a significant drop downstream. Heat transfer levels were found to be greater near the leading edge compared to the mid-gap region. The higher film effectiveness values near the suction corner were explained to be due to endwall secondary flow, which was sweeping away the coolant from the pressure surface towards the suction surface. This is clearly a three-dimensional behavior of the flow.

From [5] it is apparent that data obtained using flat-plate geometry will not be sufficient to resolve the three dimensionality of the flow near the endwall region. The importance of capturing three dimensional effects is critical given the fact that endwall losses can add up to 1/3 of the overall turbine loss [6], and therefore they need to be predicted accurately. The influence of the endwall on the overall vane heat transfer becomes more significant due to the smaller aspect ratio designs for modern gas turbines. In general, cascade data are closer to reality, but lack realistic rotational effects, and the actual operating conditions. In addition to capturing three-dimensional effects, cascade
setups also give information on the effects of curved surfaces of airfoils that may significantly influence film-cooling performance. It is known that the convex shape of the suction surface stabilizes the flow, whereas the concave shape of the pressure surface has a destabilizing effect. The convex shape has an effect of pushing the coolant towards the surface, increasing effectiveness. Therefore, while a double row of holes would be enough on the suction surface to prevent lift-off, the pressure surface needs more rows of holes for an effective cooling. Schwarz et al. [23] studied curvature effects on film effectiveness. By combining their results with those of previous studies of Schwarz and Goldstein [24], Ito et al. [25], and Pedersen et al. [26], they concluded that a convex surface has improved cooling effectiveness at low momentum flux ratios whereas a concave surface has higher effectiveness at high momentum flux ratios. The flat surface results of [25] showed a similar trend to convex surface results of [23], but produced a lower level of effectiveness.

In order to simulate realistic upstream boundary conditions for downstream coolants in a turbine vane, Polanka et al. [27] examined a six-row showerhead configuration and observed its effects on downstream cooling. It was concluded that showerhead cooling was decreasing effectiveness along the pressure surface at low blowing ratios, but at high blowing ratios its primary effect was to increase turbulence levels with associated increase in effectiveness. Colban et al. [28] measured heat transfer coefficients and adiabatic effectiveness in a two-passage scaled vane cascade with fan-shaped cooling holes. Their results showed that the existence of a showerhead-cooling region helped to reduce jet lift-off and increased adiabatic effectiveness on the pressure surface. They also provided a comparison to the vane data of Polanka et al. [27] with cylindrical holes, and the flat plate data of Gritsch et al. [16]. This comparison showed that fan-shaped holes increase adiabatic effectiveness significantly over cylindrical holes, and the flat plate data agreed very well with the suction surface of their vane data. More recently, Kost and Mullaert [29] studied film cooling in a transonic linear cascade consisting of ten blades. They traced the migration of the coolant in the endwall due to secondary flows using a laser velocimetry technique, which determined the coolant concentration and identified the ineffective cooling holes on the endwall.
Another way of simulating proper upstream conditions is to introduce radial
temperature distortion onto the inlet temperature profile, so that the influence of the
combustor temperature profile on the flow characteristics could be mimicked more
closely. Hot streaks can be positioned to impinge upon or miss the vane leading edge, and
the local heat transfer is affected differently depending on this alignment, resulting in
potential benefits regarding component life. Jenkins et al. [30] investigated the effects of
high turbulence and film cooling on hot streak strength. They used a linear vane cascade
for their experiment. Both high turbulence and film cooling were found to be decreasing
the hot streak strength with a combined effect of up to 70% reduction. Amongst the three
cooled regions they studied, the suction surface was the most effective region in reducing
the hot streak strength over the pressure surface and showerhead regions. They attributed
this result to the differences in cooling characteristics and the local hot streak thickness.
Using the same cascade, Varadarajan and Bogard [31] observed that hot streaks caused
significant changes in adiabatic effectiveness in the regions of film cooling. They also
reported sharp gradients in the temperature levels on the suction surface due to hot
streaks.

Rotation is responsible for Coriolis and centrifugal forces acting on the coolant
passages. Centrifugal force may cause a change in density gradient in addition to the
body force it is creating, which results in buoyancy forces. However, Elovic and Koffel
[32] suggest that buoyancy forces are usually negligible in high-pressure turbine blade
cooling passages. Coriolis forces give rise to secondary forces in cooling passages by
affecting internal pressure gradients. Many studies investigating internal cooling
techniques can be found in the review by Han et al. [33].

Wake simulation has been the approach utilized in stationary cascade setups to
investigate the vane wake and/or rotation effects on the downstream blades. One of these
studies is that of Heidmann et al. [34] who used a stationary annular blade row and an
upstream rotating rod wake generator. They measured the time-averaged film
effectiveness distribution, and observed that with an increase in rotational speed the film
effectiveness was decreasing on both sides of the airfoil with a more significant decrease
on the suction surface. It was suggested that this decrease may be due to the wake-
induced mixing of the coolant with hot gas path. Ekkad et al. [35] measured the combined effect of turbulence and unsteady wakes via two turbulence grids and a rotating wake generator. The two effects, when combined, produced higher heat-transfer coefficients, and when the turbulence was stronger, the effect of unsteady wake was not as significant.

The unsteady effects are not only caused by the upstream wakes, but for the case of a transonic turbine the shock waves emerging from the upstream vane row become another unsteadiness source. The shock waves induced by the trailing edge of the vanes propagate to the downstream blade row and subsequent reflections of the shock wave system propagate back to the vane row influencing the pressure distribution on the vane suction surface, and therefore leading to potential variations in the coolant blowing rates as well. For example, in another wake simulation study, Rigby et al. [36] investigated the shock wave effects on the downstream blade row in addition to the wake effects. They found that these effects influenced film-cooling behavior significantly. The wake clearly enhanced mixing of the coolant on the blade suction surface, but the pressure surface was hardly affected by these effects, mostly due to coolant lift-off even at low blowing ratios. The presence of shock waves can produce variations in unsteady heat transfer rates. Popp et al. [37] investigated the unsteady heat transfer caused by the shock wave using a five-passage cascade, and a shock tube to create the shock waves. They concluded that the major contribution to the unsteady heat transfer was due to the temperature fluctuations induced by the passing shock wave.

3.1.3 Rotating Turbine Data Base

Despite the vast amount of effort spent in this topic area, one clear observation is that cascade experiments deal with stationary airfoils, and rotational and unsteady effects can only be generated up to a certain extent. Rotating turbine configurations provide more realistic data, because they represent the real engine environment as closely as can be done under controlled laboratory conditions by matching non-dimensional parameters at operating conditions. However, to this point in time there is very little film cooling work done using full-scale turbine rotating rigs, mainly due to difficulties of direct measurement of film effectiveness. An in-depth review of literature regarding turbine
heat transfer and aerodynamics has been recently performed by Dunn [20], which also includes all existing rotating rig experimentation up to 2001.

Rotating rig experiments are performed in facilities known as short-duration or long-duration. The blowdown and shock-tunnel experiments are capable of performing experiments in the order of milliseconds, which is the reason why these facilities are called short-duration. The long-duration facilities utilize large-scale models, and require a large amount of power. They operate at low speeds, and cannot match the operating parameters. There are currently two major short-duration facilities in the United States: the shock tunnel facility at OSU GTL pioneered by Dunn during his days at Calspan Corporation [38], and the blowdown type Turbine Research Facility (TRF) at Wright Patterson Air Force Base (WPAFB), construction of which was initially overseen by Dunn and Haldeman [39] in conjunction with Epstein and Guenette from MIT. There is also the blowdown facility at MIT pioneered by Epstein and Guenette that was used for rotating rig experimentations in the 1990’s [40]. A long-duration facility that was used as in the early stages of this type of work is the Large Scale Rotating Rig (LSRR) of United Technologies Research Center (UTRC) [41]. One of the currently operational long-duration facilities is the Axial Flow Turbine Research Facility, AFTRF, at Penn State [42]. There is also a long-duration facility -Warm Core Turbine Test Facility- at NASA - Glenn Research Center [43]. In addition to the ones in the US, there are similar facilities globally. At Oxford University, there is the Isentropic Light Piston Tunnel (ILPT) pioneered by Schultz [12], another type of short-duration facility that utilizes an isentropic light piston tube to create the desired inlet flow conditions. The light piston facility of Von Karman Institute (VKI) [44] has been mainly used for aerodynamics measurements. The QinetiQ Isentropic Light Piston Facility [45] is a more recent short-duration facility, concentrating on heat-transfer measurements. The Turbine Test Facility-LISA- of Swiss Federal Institute of Technology (ETHZ) [46] is an example of a long-duration facility.

Rotating rigs have the potential to provide controlled laboratory measurements for flow more closely representing corrected engine conditions than other experimental configurations. Some of them, however, cannot produce the engine corrected flow
conditions. For example, long-duration (low speed) rigs are continuously run for hours at a time, but cannot reproduce the design corrected flow conditions. They are accepted to be providing adiabatic conditions, because there is almost no heat transfer to the surrounding metal after hours of running. As noted, many of these long-duration rotating facilities are capable of measuring rotational effects, but they cannot match the non-dimensional parameters of a real engine, such as the pressure ratio or flow function. The short-duration rigs, on the other hand, can reproduce the design corrected conditions associated with the turbine stage, and the aerodynamic and heat-transfer measurements are captured in an isothermal environment.

### 3.1.3.1 Rotating Rig Experiments Without Cooling

Dunn and his coworkers performed many of the rotating rig heat-transfer experimental results reported in the open literature using short-duration experimental facilities. The shock tunnel currently being used at The Ohio State University, which was used extensively in shock mode in the past, is the facility used in this current research and is now being run in blowdown mode in order to be able to produce the various vane inlet temperature profiles. While both modes are capable of capturing aerodynamics data, shock mode was preferred more in the past for heat-transfer measurements. The reason behind this is that turbine inlet temperatures as high as 600K could be achieved within much shorter time, such as 25-30 ms, in the shock mode, via a reflected shock wave created upstream of the test rig. This temperature difference between hot mainstream gas and the metal surface is critical for the accuracy of heat-transfer measurements. When operating in blowdown mode, the inlet temperature stays much closer to room temperature (around 300K) unless a method is introduced for heating the gas. More recently, a resistance heater that is capable of producing temperature profiles at the high-pressure turbine vane inlet consistent with those coming from an engine combustor has been incorporated into this test facility upstream of the vane row, enabling heat-transfer measurements in the blowdown mode as well. The run time in the shock mode depends on the combination of the tube length and diameter. For the blowdown mode, the run time primarily depends on the volume of the dump tank necessary to sustain choked flow.
at the turbine exit and that volume has recently been increased significantly. The typical blowdown mode run time is on the order of 300-400 ms.

Guenette et al. of MIT [70] collected time-accurate heat-flux measurements on the rotor blade mid-span using thin-film heat-flux gauges. A coherent disturbance along the pressure surface, and the blade passing periodicity in the ensemble-averaged data was observed. This is consistent with the phase-resolved results of Dunn et al. [47]. Polanka et al. [48] performed tip and shroud heat-transfer measurements using the facility at WPAFB, and investigated the unsteady effects. The peak heat-transfer values were occurring as the blade passed over the gauges located on the shroud. Later, Thorpe and Ainsworth [49] performed more detailed measurements in the blade tip-region of a fully scaled transonic turbine stage in ILPT facility of Oxford University. This data set, consisting of heat transfer coefficient and adiabatic wall temperature, targeted to separate these two effects on the fluctuating heat transfer rate, and showed that the maximum heat transfer coefficient was occurring when a blade tip passed over the measurement point, which is parallel to the findings of Polanka et al. [48].

Radial temperature distortion is another topic that has been investigated in rotating rig environment. Butler et al. [50], Shang and Epstein [51], and Roback and Dring [41] are among the researchers who performed rotating rig experiments to analyze the hot streak impact on the downstream rotor. They all observed that the effect of hot streaks made it through the first stage vane to still have an impact on the downstream rotor. Roback and Dring [41] also found that there was no difference occurring at the rotor leading edge due to the alignment of the hot streaks either with vane leading edge or with mid-pitch. This was in contrast to the numerical results obtained by Gundy-Burlet and Dorney [52] that showed if hot streaks were aligned with vane leading edge, their effect on the downstream rotor decreased due to the deceleration and mixing taking place in the flow field. Butler et al. [50] reported that the migration of hotter gas onto the pressure surface of the rotor due to the hot streaks caused a significant change in the rotor field. Shang and Epstein [51] showed that the radial temperature distortion caused about a 10-30% increase in heat transfer levels on the rotor, mainly on the pressure surface, compared to a uniform profile.
More recent research has started focusing on the effects of the temperature distortion on the first stage vane in the rotating rigs. Povey et al. [53] investigated the effect of hot streaks on the HP vane surface and endwall heat transfer. The QinetiQ Isentropic Light Piston Facility mentioned above was used for these experiments. They reported a significant increase at the heat transfer rate on the mid-span suction surface when the hot streaks were aligned with the vane leading edge, and a slightly less increase when aligned with the mid-passage, compared to the uniform profile effects. However, no significant difference was observed on the pressure surface for these three profiles. At the vane endwall (both hub and casing), they observed higher heat transfer with uniform profile, and the alignment of hot streaks did not create a difference in this region. An increase in heat transfer rate through the passage was apparent with an increase in axial chord. Barringer et al. [54, 55] also studied the combustor exit profile shapes on an HP vane outer endwall. This was a vane-only configuration, and the WPAFB blowdown test facility was utilized for the experiments. Differences in the inlet pressure profile were found to be causing significant changes in the secondary flows affecting larger areas within the vane passage. Also, their findings showed parallel results to those of Povey et al. [53]: the uniform profile showed higher heat transfer rate at the endwall compared to the distorted temperature profiles, and an increase was observed with the axial distance. They concluded that heat transfer varies as a function of inlet temperature profile and the location at the endwall. At this point, the author wants to draw attention to the fact that [53] and [54-55] offer very rare examples of detailed work concentrating on the first stage vane (rather than the rotor) heat transfer in the rotational environment. However, both efforts lack the influence of film cooling.

The shock tunnel facility used in this current research was originally designed, constructed, and used at Calspan Corporation, and later was moved to OSU. Dunn et al. [38] provide a detailed description of the facility, and related information can be found in literature such as in Dunn et al. [56-59], and Bergholz et al. [60].

Dunn et al. [61] collected heat flux data at the endwalls and the meanline of a nozzle guide vane of the Garrett low-aspect-ratio (LART) stage. The Stanton Numbers were found to be slightly increasing towards the trailing edge, which may be due to
acceleration of the flow in this region. Via the predictions, they concluded that the pressure surface boundary layer of the vane was turbulent starting from the beginning, while the suction surface went through transition very soon behind the leading edge. Another research program of Dunn et al. [62] investigated the vane/blade spacing influence on the unsteady heat flux of vane and blade mid-spans of the Allison Vane-Blade Interaction (VBI) turbine, and drew the same conclusion that vane/blade spacing affected neither the vane nor the blade. Haldeman and Dunn [63] reported heat transfer measurements with predictions at different spans on both vane and the blade, inner and outer endwalls, and the blade tip and shroud. This was a data set accompanying the pressure measurements and predictions performed by Bergholz et al. [60] as a part of the same research program. The heat transfer through the passage was found to be increasing with axial chord, with pressure and suction surfaces reaching the highest levels compared to the mid-passage. Vane endwall was observed to have similar heat transfer levels to those on the airfoil surfaces. The results also showed that there was a variation both on the vane and the blade as a function of span due to the 3-D characteristics of the flow.

A more recent research program that has been ongoing using the same facility is the GE USA program, which was mentioned previously. The initial stage of this program (Build 1) concentrated on time-averaged and time-accurate aerodynamics measurements. Green et al. [64] analyzed the aerodynamics of a high-pressure turbine blade with a recessed (squealer) and a flat tip, and obtained good correlations with unsteady CFD predictions. Another focus was on vane clocking effects on aeroperformance. Clocking is aligning one vane (blade) row with the wake of another in order to decrease the viscous effects by the help of upstream low momentum wake. Haldeman et al. [65] observed the effects of low-pressure turbine vane clocking on overall turbine performance. They concluded that due to the strong disturbances introduced by the coolant and the combustor, clocking effects would not be significant in a fully-cooled engine. However, even if the overall machine performance was not affected, the inlet static pressure condition of the low-pressure turbine was affected, and this could influence the cooling performance of the machine. Details on this study can be found in Haldeman [66].
Both of these studies obtained data in blowdown mode, as the content was concentrating on aerodynamics only. Haldeman et al. [67], demonstrated a combination of heat transfer (from shock mode) and aerodynamics (from blowdown mode) data, and showed that the results were insensitive to the operation mode of the facility. Tallman [68], Tallman et al. [69], and Molter et al. [70] performed predictions for comparison with this heat-transfer data set concentrating on different regions of the blade.

3.1.3.2 Rotating Rig Experiments With Cooling

Film cooling experiments in rotating rigs are few in number. Dring et al. [71] were the earliest and one of few researchers, who performed film-cooling study for a rotating rig. They studied a rotor blade of a large-scale model, with two cooling holes (a modern vane might include 300 to 400 cooling holes) located at mid-span suction and pressure surfaces, with a range of blowing and density ratios. They compared their data with previous flat plate and cascade data. Their findings showed that suction surface film cooling behavior was similar to the non-rotational cases, but pressure surface was experiencing a faster decay in effectiveness due to curvature and radial flow.

Dunn [72] performed measurements of heat-flux distributions for a Garrett TFE 731-2 HP turbine with slot injection at the NGV trailing edge using the current facility, and observed that the presence of the rotor increased the heat transfer on the vane trailing edge by up to 25%, but no significant effect was observed for the rest of the mid-span. Dunn [73] also showed, for the same turbine stage, that the upstream NGV injection significantly increased the downstream blade leading edge heat flux. Later, Metzger et al. [74] performed time-averaged predictions on this data. However, in another experiment performed for the Teledyne CAE 702 HP turbine, Dunn and Chupp [75] observed that the vane injection had no influence on blade heat flux. They attributed this to the differences in the turbine aerodynamics and the injection techniques used for the two stages. They also concluded that the vane/blade spacing did not affect the vane and blade heat transfer.

Later, Takeishi et al. [76] measured film-cooling effectiveness of a high-speed rotating rig and a low-speed linear cascade; both for the same scaled model blades, and compared the results of the two. The blade they used had three rows of showerhead cooling, two rows at pressure and one row at suction surfaces. However, the actual
density ratio was not matched, and there were limited data points on the pressure surface due to measurement difficulties. Their results were still in parallel to what Dring et al. [71] observed: The film cooling effectiveness in the leading edge region of pressure surface was decaying more rapidly than that of the cascade. They concluded that this was due to the radial flow effects and high mixing occurring on the pressure surface.

Abhari and Epstein [77] obtained unsteady heat-transfer measurements on a film-cooled blade of a transonic rotating turbine stage. They compared the results of the cooled rotor to the un-cooled rotor and to the cooled linear cascade of the same profile of Rigby et al. [36]. The results suggested an effective film cooling on the suction surface that compared favorably to both cooled cascade and un-cooled rotor cases, but little reduction in heat transfer along the pressure surface was obtained. High blowing ratios were observed to decrease effectiveness, being in agreement with previous studies. Heat transfer was affected by the unsteadiness caused by blade row and wake-blade interactions, especially near the front of the blade. This resulted in an unsteady coolant performance.

Some of the experiments investigating the secondary cooling effects such as platform cooling and purge flow cooling on the rotor have mainly focused on the aerodynamic performance outcomes. McLean et al. [78] showed that this small amount actually changed parameters such as pressure coefficient and exit angles of the turbine stage, significantly. This indicated that the endwall secondary flow was reshaped due to the changes in the rotor inlet boundary layer caused by the purge flow. The strongest effects were observed in the region from the hub to the mid-span. Pau et al. [79] also observed that the purge flow was responsible for enhancing the rotor endwall secondary flow.

None of these studies had film cooling on the vane, but only a few studies included slot injection [72,75] that is immediately effective on the vane itself. The study of Haldeman et al. [1-2] reports fully-cooled turbine stage steady and unsteady measurements, where both the vane and the blade were fully cooled. For these experiments, the vane inlet temperature profiles and both the vane and the blade blowing ratios were varied, while the design corrected conditions were matched. Blade heat-
transfer measurements were performed at several different span locations and in the recessed tip region, but the vane was instrumented only at mid-span. The heat transfer on the vane pressure surface was observed to be lower than that measured for the suction surface in part due to the addition of significant cooling gas on the pressure surface. Film cooling had more significant effect on the blade than on the vane. The trend of heat transfer was found to be differing for mid-span and for tip region of the blade, on the pressure surface toward the trailing edge. The influence of unsteady flow on cooling was also evident in these measurements. Following these measurements, Southworth et al. [80] and Haldeman et al. [81] performed the comparison of time-accurate predictions with this data.

The most recent study is the GE USA program Build 2 Entry 1 rotor heat transfer analysis performed by Mathison [82], which is the accompanying work to this thesis. The rotor was un-cooled, but received cooling effects from the upstream cooled vane and the purge flow through the wheel-space cavity. The influence of temperature profiles and cooling levels including the purge flow on the un-cooled rotor was examined. The temperature profile effects were found to be more dominant over cooling flow effects, among which the uniform profile showed the cooling effects more clearly. The vane outer cooling was found to have the greatest influence on the rotor heat transfer, and the vane inner cooling had the least effect. Cooling had more impact on the suction surface compared to the pressure surface of the rotor blades. The predictions obtained via FINE/Turbo for the un-cooled runs of Build 2 with the isothermal wall boundary conditions gave satisfactory match with blade leading edge and platform flow field temperature data. The detailed analysis regarding the effects of the cooling variation, temperature profiles, vane outer cooling and purge flow on the blade row heat transfer, and the computational effort can also be found in references [9, 83-86].

As observed so far, there is extensive research in the area of film cooling, but not all of the results are necessarily applicable to gas turbine design due to the lack of similarity to actual engine environment. The measurements of Haldeman et al. [1-2] provide the most recent data set in today’s literature, but the main focus on that study was on the blade, and the vane heat-transfer measurements were performed only at mid-span.
Although the vane was fully cooled, the endwall cooling was not of high coverage and was not instrumented for those experiments.

Current research is a part of the most recent stage of the GE USA research program mentioned above, which concentrates on heat transfer measurements on a fully film-cooled high-pressure turbine vane row. The measurements are not confined to the mid-span, but are obtained at multiple spans and inner endwall with a better cooling coverage, while the non-dimensional engine parameters are matched.

Although the experimental measurements performed for rotating rig environment have been the main focus of this section, these experimental programs were mostly accompanied with numerical predictions utilizing various CFD codes. These predictions can also be found in the references mentioned above. Next, the effort invested in improving these prediction capabilities for the film-cooling influence will be described.

3.2 Computational Work

There are currently various types of CFD design tools used by the turbine design community. More recent ones make use of Navier-Stokes equations, and they are capable of 3-D solutions as well as handling unsteady effects. Managing successful designs has been a motivation in CFD tool development, which is today driven towards ability of more complex and multi-stage cooled turbine simulations. Advances in these prediction techniques result in more accurate solutions, but requiring large amount of computational resources at the same time. Therefore, a designer will often use a simpler tool in order to obtain timely results, when accuracy is not the high priority on the list. Even if there is also a well-documented source of cascade data in the literature, such as the NASA HOST (Hot Section Technology Program) data base [87-88], these predictions are mainly based on boundary layer models, which utilize flat-plate geometry and basic heat transfer correlations.

It is not surprising that these models cannot capture the engine environment accurately, as the flow physics proves to be highly complex and three-dimensional in the engine environment. In addition, the correlations can only reflect the design features that were used at the time the codes were developed, and there is a time lag between the experimental investigation and the code verification, as mentioned earlier.
In addition to the specified inlet and exit flow parameters, the pressure and velocity distributions are also required for heat-transfer calculations, and these are the output of the aerodynamic calculations. Therefore, an accurate aerodynamics solution is necessary to be able to obtain a reliable heat-transfer solution. The grid to be used for the heat-transfer calculation needs to be much finer due to the temperature gradient at the wall. Due to these difficulties, the current state-of-the-art is that aerodynamics tools have seen more improvements compared to the heat-transfer tools. In addition, regardless of computation being heat transfer or aerodynamics, in order to obtain a full simulation the vane endwalls, the blade platforms, and tips need to be included in the modeling.

There are many computational studies that have been performed using either aerodynamic or heat-transfer design tools. The recent study of Tallman [68], reports the development of the GE proprietary three-dimensional CFD solver TACOMA to be able to perform both aerodynamics and heat-transfer computations, which was confirmed successfully by using the experimental data of Haldeman et al. [67] collected at realistic engine conditions for the un-cooled turbine stage. This methodology should be utilized for further improvements in heat-transfer predictions to also account for film cooling modeling.

There are several different approaches to the prediction of the influence of film cooling on the local heat transfer. One approach is the introduction of a jet injection model into the design codes. The coolant jet and the boundary layer interaction are modeled via the mass, momentum, and energy balances. The early work was initiated by implementation of the jet model into two-dimensional boundary layer codes. Crawford et al. [89] modified STAN5, a two-dimensional boundary layer code, with the addition of the subroutines containing the injection and the turbulence augmentation models. The resulting program, STANCOOL, had limited success without the effects of rotation and curvature. Schönung and Rodi [90] proposed a more complex injection model to account for lateral mixing. The model simulated the film cooling effects via a three-dimensional elliptic flow, and a new boundary layer was specified after this near-hole elliptic region. Tafti and Yavuzkurt [91] accounted for the three-dimensional mixing of the coolant in the Crawford model, and the two-dimensional row injection was predicted successfully.
There have been many other efforts by researchers to improve these existing injection models for better predictions.

Later, Abhari [92] extended the model of Tafti and Yavuzkurt [91] and incorporated it into the two-dimensional unsteady Navier-Stokes code UNSFLO. The predictions were compared to the VKI film-cooled linear cascade data of Camci [93], and to the short-duration film-cooled rotating rig data of Abhari and Epstein [77] obtained using a Rolls-Royce ACE turbine as the test article. The predictions were successful except in the immediate vicinity of the hole. Garg and Gaugler [94-95] modified a three-dimensional steady Navier-Stokes code to include film-cooling effects. The experimental data used for the comparisons were the C3X vane of Hylton et al. [96], the VKI blade of Camci and Arts [97], and the ACE blade of Norton et al. [98], which were all stationary, and satisfactory results were obtained.

The common practice in industry has been to add the coolant into the domain directly through the use of source terms due to its practicality. However, since not all the regions around the airfoil surface have a good resolution (such as the tip gap region), individual cooling holes cannot be adequately modeled. Therefore, the cooling flow is spread among numerous cells in the computational domain either as lumped mass, or distributed mass.

Instead of implementing a jet model into the CFD solver, a more idealistic method of simulating cooling flow is by directly solving for the flow field. Leylek and Zerkle [99] argued that a two-dimensional boundary layer model needs to be calibrated with a database to produce reliable results, and, therefore, it cannot predict film cooling properly when used for a totally new configuration without performing a re-calibration. The approach was to compute the flow in the plenum, the film hole, and the cross-stream. Of these, the film hole is the most difficult to grid due to the complex flow nature. The strength of this complexity was found to be highly dependant on the L/D, BR, and the injection angle. Later, Walters and Leylek [100] outlined the procedure to be applied in this type of computational methodology in order to improve accuracy of predictions. A correct model of flow physics, a high-quality grid, and a higher-order discretization scheme are among the essentials.
For the sake of accuracy, this approach requires the application of an extensive meshing for the coolant holes, for the plenum chamber, and the near hole region, which all have been explained to have an effect on the coolant interaction with the mainstream. It provides a more accurate and detailed solution for the cooled turbine passage, but also requires the use of millions of grid points within the mesh. Hence, one needs both time and a sufficiently large amount of resources to handle this type of heavy computation. When it is considered that modern HPT vanes and blades may contain hundreds of cooling holes, the growing complexity of these models represents a major barrier to accurate prediction of cooling performance.

With the motivation of lessening this computational burden while still obtaining accurate results, Burdet et al. [101] extended the aforementioned two-dimensional jet injection models to a macro-model approach in order to improve the film-cooling modeling concept. This type of model is a hybrid approach to the simulation of mixing region near the cooling holes instead of gridding all cooling holes with millions of grid points. The model accounts for the jet trajectory, the mixing of the coolant with the mainstream, its penetration through the boundary layer, the wake zone, and the secondary flows. The velocity and thermal fields are calculated via the modeling of a counter-rotating vortex pair, CVP, which is created due to the coolant jet interaction with the mainstream. In this study, MULTI3, a three-dimensional unsteady RANS CFD code was used, and was verified with the flat-plate data of Saumweber et al. [102]. Later, Burdet and Abhari [103] verified this model using the linear transonic cascade data of Schulz et al. [104]. There was a significant mesh size reduction on the order of one order of magnitude, and a significant reduction in computational time (about one or two orders of magnitude) when compared to an extensive mesh including the holes and the plenum chamber. Further, the predictions were in good agreement with the data.

It should be noted that the references given in this section are selected to be among those that introduced a new approach to film-cooling modeling in the computational area. In addition to these efforts, there have been many other computational studies targeting the predictions at various regions on film-cooled airfoil surfaces. Computations have been performed for internal cooling passages and also for
external surfaces, for film-cooled cascades, as well as for film-cooled rotating machinery as was mentioned before. As computation accuracy has become more and more important in this competitive industry, the accuracy of the turbulence models used in design tools has also drawn more interest.

Both the experimental and computational portions of the literature covered here only scratch the surface of the vast research in the film cooling area. The reader interested in further details on this subject can find many reviews in references. One of the earliest reviews of advances in film cooling area was performed by Goldstein [105], which covers the discrete film cooling papers appearing in the literature for the period until 1971. Beyond 1971 until 1998, Kercher provided a review for film cooling at the leading edge [106], and film cooling CFD efforts [107]. Elovic and Koffel [32] presented a review on the state-of-the-art (1983) design of turbine cooling systems. Simoneau and Simon [108] discuss the turbine gas path heat-transfer experiments and computational work in cascades and rotating rigs performed until 1993. Later, Bunker [109] presented a review on turbine blade tip heat transfer in 2000. Dunn [20] has written a thorough review of heat transfer and aerodynamics in turbines, both in experimental and computational areas covering both cooled and un-cooled designs in 2001. In a recent VKI lecture series (2007) [110], the state-of-the-art experimental and computational film cooling technology was reviewed. These reviews form a valuable source of literature survey on this subject.
CHAPTER 4

EXPERIMENTAL METHODOLOGY

In this chapter, the experimental setup, data collection and data processing procedure for the high-pressure turbine film cooling experimental series used for this thesis work will be explained. The aim is to familiarize the reader with the sequence of steps in the experimental plan used to obtain the raw data. The shock tunnel facility and its components used to simulate the working environment of a film-cooled high-pressure turbine are explained in detail. The data collection system used for data recording during the experimentation is introduced, and some sample measurements used to obtain the turbine experimental operating conditions are presented. Finally, the calibration and processing techniques will be discussed.

4.1 Test Facility

The function of the test facility is to provide the desired flow conditions at the inlet to the turbine stage. For the experimental series of interest to this thesis, the shock tunnel operated in blowdown mode to supply the desired inlet pressures, while the large cooling facility (LCF) supplied cooling flows, and the combustor emulator created the prescribed temperature profiles at the vane inlet.

4.1.1 Shock Tunnel Facility

The facility consists of two sections: the driver and the driven tubes that are 40 ft and 60 ft in length, respectively. A Mach 6 expansion nozzle is located at the exit of the driven tube to tailor the total pressure downstream of the shock wave to the desired inlet condition, and opens into a dump tank of 9 ft diameter and 33-ft length (now 45-ft in length), inside of which the instrumented rotating rig is located.

For a given driven tube initial pressure, the axial location of the rig inlet inside the expansion nozzle determines the inlet total pressure. Most of the pressurized air bypasses
the rig entrance and passes into the dump tank. A small fraction of the flow passing through the choked nozzle at the exit of the tube (generally on the order of 1/10 or 1/20) makes it through the turbine stage. The turbine stage is operated at the design flow function and thus the flow function, the inlet temperature, and the inlet pressure set the mass flow rate through the machine.

When operating in blowdown mode, the 100-ft of tube provides the reservoir that is pressurized to a predetermined value via an external pressure source. As noted above, a heater (known as a combustor emulator) is placed upstream of the high-pressure vane inlet in order to heat the gas and to provide the desired temperature profile prior to passing through the turbine stage. A view of the shock tunnel from the Gas Turbine Lab is given in Figure 4.1.

![Figure 4.1 A View of the Turbine Facility](image)

Prior to initiation of the flow into the turbine stage, the dump tank where the rotating rig resides is evacuated to a pressure on the order of 2-torr to reduce air resistance so that the turbine disk can be accelerated to the desired pre-run speed in a
minimum time. In addition, evacuation of the dump tank is also necessary to allow for flow establishment and to sustain the desired stage pressure ratio for a lengthy period of time. Prior to initiation of the experiment, the large cooling facility (LCF) and the combustor emulator are brought to proper temperatures. Depending on the temperature profile, it may take several hours for the heater to achieve the desired conditions. After the driver tube is pressurized to the desired level, the turbine is brought up to about 1% below the desired physical speed necessary to duplicate the design corrected speed during the experiment. The air supply to the motor driving the turbine is terminated at the time the main flow valve is opened so that the turbine is free to accelerate as work is extracted from the flow. The experiment is initiated when the operator pushes a manual trigger that controls operation of the coolant flow and the mainstream flow. First, the LCF valve opens so that the cooling gas supplied to the turbine stage can be stabilized prior to initiation of the mainstream flow. Later, the FAV opens to initiate the mainstream flow. The data collection system is initiated prior to the time the coolant and the pressured air are supplied to the turbine stage so that data can be collected at corrected design conditions. The supply lines and cooling circuits for the rig configuration can be seen in Figure 4.2. As previously noted, the rotor speed increases during the experiment (on the order of 2%) because the turbine is extracting work from the fluid flow. The torque produced is measured by the known time rate of change of speed and the moment of inertia of the rotating system. An alternative method used in some other facilities is to introduce an eddy brake to hold the speed of the turbine constant, and the torque produced is measured via the eddy brake. The difficulty with this approach is that it limits the available slip ring channels. The fast-acting valve stays open for about 125 ms, followed by the closing of LCF valve. Opening another valve and flooding the test section with air in order to slow down the rotor is the approach used to terminate the experiment. This flow of the experiment will be explained in detail in Section 4.5 with accompanying figures.
The valve separating the supply from the test section can be held open for a time period selected by the operator making the run time for facility operation in blowdown mode longer than when operating in shock-tube mode. More importantly, by using the combustor emulator upstream of the vane inlet it is possible to generate temperature profiles, making this mode of operation very desirable for film cooling experiments. The temperature profile that is obtained at the vane inlet when operating in shock-tube mode is uniform.

4.1.2 Large Cooling Facility (LCF)

The coolant gas used in the experiments is dry air supplied by the LCF at about 260K at a pressure that the operator can select in the range of 40 to 120 psia. The supply temperature is controlled via a chiller mounted on top of the tank as shown in Figure 4.3.
After passing through venturi-style metering chokes, the coolant is distributed into three cooling circuits; the vane inner, vane outer, and purge cavity. After the coolant reaches the dump tank, the coolant in the vane inner and purge circuits enter the rig through separate struts in order to reach the corresponding cooling plenum. The purge coolant is supplied to the purge cavity, while the inner vane coolant is distributed through the holes located on the inner wall of the vane. The outer vane coolant is supplied directly from outside of the rig.

The coolant mass flow at each circuit is obtained via the measurements of choke effective areas, measured temperatures, and measured pressures for each cooling circuit. This information is also obtained from the pressure decay rate in the LCF tank, for consistency and leak checks. Therefore, the overall mass flow rate is known through each circuit, and this mass is distributed among the individual holes on the vane. The coolant initially chokes at the cooling holes on the vane surface, as well as on the venturi-type chokes in each of the coolant lines, when the dump tank is still at vacuum just prior to opening of the main flow valve that initiates the external flow. Once the main flow starts the cooling holes un-choke. Therefore, the mass flow through individual holes during the useful time is dependant upon the specific characteristics of each individual hole.
Different coolant blowing ratios are obtained by varying the mass flow rate, which is easily done by changing the coolant tank supply pressure. The blowing ratio for the individual rows of cooling holes is determined by flow area, internal coolant supply conditions, and local external flow parameters.

For a compressible flow of an ideal gas where the isentropic relations hold, the following relations apply for each cooling circuit:

\[
\dot{m} = \rho U A = \frac{P}{RT} AM = \rho AM \sqrt{\frac{\gamma}{RT}} \tag{14}
\]

\[
M = \sqrt{\frac{2}{\gamma-1}} \frac{T_0}{T} \tag{15}, \quad \text{and} \quad \frac{T_0}{T} = \left(\frac{P_0}{P}\right)^{\frac{\gamma-1}{\gamma}} \tag{16}.
\]

\( \dot{m}_{\text{actual}} \) is calculated through equation (14), with the information of the measured area from a blowdown. From \( \dot{m}_{\text{actual}} \), blowing ratio, BR, is calculated:

\[
BR = \frac{\rho U_c}{\rho_c U_c} = \frac{\dot{m}_{\text{actual}}/A}{\rho_c U_c} \tag{6},
\]

where A is the cooling circuit area.

Standard procedure is to bring the LCF to the proper coolant temperature before the experiment. At the onset of the experimental series, this can take up to 36 hours if started from room temperature, but once the desiccant is completely dry and the tanks are thoroughly cooled, the system maintains its temperature for many weeks.

4.1.3 Combustor Emulator

The combustor emulator used in this research was designed and constructed at the OSU GTL. Its working mechanism uses a passive heat exchanger technique, where the working fluid (air) is passed through a honeycomb heater matrix constructed of Inconel, and the temperature profile of the matrix is transferred to the fluid. This temperature profile is maintained as the fluid propagates downstream to the high-pressure turbine vane inlet. The system allows for different temperature profiles to be created, with high peak and bulk temperatures, so that the temperature profile consistent with that
experienced in a real engine environment can be replicated more accurately. The number and pitch location of hot streaks can be changed easily by relocating the heater rods inside the matrix.

Figure 4.4 Combustor Emulator – Upstream View (on the Left) and From Another Angle (on the Right)

The left portion of Figure 4.4 is an upstream view of the combustor emulator while the right side provides a view of the overall assembly. The turbine stage is located behind the emulator. The metal circumferential layers seen at the front serve as a protective layer to the honeycomb matrix located behind, so that the initial impulse load associated with the system flow establishment process does not destroy this structure, but instead enters the section smoothly. They are also utilized as channels to effectively shield the wires coming out of the emulator to the external power supply. This metal section is not needed on the other side of the emulator, and the flow goes into the turbine stage flow path directly through the honeycomb matrix.

The desired temperature profile is created in the combustor emulator before the experiment sequence is initiated. The power density of the heaters and the thermal conductivity of the material are the determining factors in the profiles. Rod heaters inserted into the matrix are used for producing these profiles, and RTD’s (Resistance Temperature Device) are placed strategically to have a measure of the local temperature
within the matrix. For the purpose of the measurement program described here, there are three temperature profiles of interest: uniform, radial, and hot streaks. Of these, radial and hot streak profiles are more interesting to the turbine designer, since the combustor exit temperature profile in an engine is generally not uniform. RTD’s are distributed circumferentially within all eight of the emulator radial rows, because a circumferential distribution is felt to be a better characterization way for uniform and radial profile cases. The interested reader can find more information on this particular combustor emulator in Haldeman et al. [111].

![Figure 4.5 A Close View of Combustor Emulator](image)

When creating a uniform temperature profile, it is necessary to utilize all of the heater rods installed in the combustor emulator, while only a portion of the heaters is utilized to create the radial or hot streak profiles. Two different sets of 19 heater rods are used to create hot streaks. One set is designed to align the hot streaks with the leading edge of every other vane airfoil, and the other set is designed to place the hot streaks at mid-passage of those airfoils, which gives a measure of the alignment effects of the fuel nozzles with respect to the vane leading edge. In order to create a more circumferentially
uniform profile, both sets are used for radial and uniform cases. For uniform profiles, there are also additional inner and outer rings of heater rods used. The honeycomb matrix structure of the combustor emulator is shown in Figure 4.5.

A radial profile is approximately a parabolic shape of temperature distribution along the span, whereas the uniform profile is nearly constant. That is, the radial profile is circumferentially uniform and spanwise radial, and the uniform profile is both circumferentially and spanwise uniform. The peak temperature of the radial profile shifts a little towards the tip due to rotation. The degree to which the uniform and radial temperature profiles can be achieved will be demonstrated in Chapter 5. Hot streaks are rather easy to create, and their alignment relative to the vane leading edge will be explained further in Section 4.3.

In summary, the combustor emulator used in the experiments is capable of providing different shape (or profile) factors. Chana et al. [45] proposed the temperature distortion factors, as:

\[
\text{Overall Temperature Distortion Factor (OTDF)} = \frac{(T - T_{\text{mean}})}{(T_{\text{mean}} - T_c)} \tag{17}
\]

\[
\text{Radial Temperature Distortion Factor (RTDF)} = \frac{(T_{\text{radial}} - T_{\text{mean}})}{(T_{\text{mean}} - T_c)} \tag{18}
\]

Or another definition that can be used is:

\[
\text{Profile factor} = \frac{T_{\text{local}} - T_{\text{ave}}}{T_{\text{ave}}} \tag{19}
\]

as defined in [84], where \(T_{\text{ave}}\) is the area-weighted average temperature, and \(T_{\text{local}}\) is the local temperature. This type of definitions is useful for quantifying the temperature profiles and for comparison purposes.

4.2 Turbine Rig

The test article used is a modern high-pressure transonic one and one-half turbine stage as shown in Figure 4.6. Figure 4.7 is a sketch of the turbine rig assembly illustrating the location of the combustor emulator relative to the turbine components and the exit
choke. The turbine stage consists of a high-pressure vane row, a high-pressure blade row, and a low-pressure vane row. The high-pressure vane is film-cooled and disk cavity purge flow is provided for this experimental configuration. This thesis specifically addresses the data obtained for the film-cooled high-pressure vane row. Overall, there are 38 vane airfoils for each vane row, and 72 blade airfoils in the rotor disk. The overall experimental configuration consists of the rig inlet, the combustor emulator, the turbine stage, and a downstream adjustable choke that is used to set the stage pressure ratio. There are two slip ring units that transfer the signal output from the rotating rotor disk instrumentation to the stationary laboratory DAS systems.

The high-pressure turbine stage pressure ratio is in excess of 5.0; and the overall stage pressure ratio across the entire turbine stage is established by the downstream variable area choke as noted above. In addition, the vane inlet total pressure (or the Reynolds number) is obtained by moving the rig inlet axially in the expansion nozzle to intercept the flow at different Mach numbers or by changing the initial supply tube pressure. The rotor torque and the corresponding power output can be determined by the
measurement of the rotor angular acceleration and the known moment of inertia of the rotating system.

Figure 4.7 Turbine Rig Assembly Showing Parts (Not to Scale)

Film cooling is managed via cooling holes located at the vane inner and outer endwall, at the leading edge with a showerhead arrangement, at the trailing edge with slots, and along both pressure and suction surfaces. There are two cooling circuits at the vane inlet cavity and the vane outer cavity. Showerhead, airfoil surfaces, and inner endwall cooling are supplied by the vane inner cavity, while trailing edge, the last three rows of holes on the pressure surface, and outer endwall are cooled via the vane outer cavity. There are six rows of holes on the pressure surface, and a set of seven rows of holes of circular type in the showerhead region, five of which are on the pressure surface. In addition to the showerhead, there are two rows of fan-shaped holes on the suction surface leading edge region. There is only one more row of holes on the suction surface of the airfoil closer to the throat area. The number of cooling holes is more on the pressure surface since the pressure surface of the vane airfoil needs more cooling coverage due to both the curvature effects and being exposed to the upstream heater. Table 4.1 gives the split of the cooling rows and the area ratios of total number of holes per circuit. According to this, most of the coolant from the vane inner circuit is used for the airfoil surface, while most of the vane outer circuit supplies coolant to the outer endwall.
Table 4.1 Cooling Holes in the Vane Section

The cooling scheme of the vane airfoil is shown in Figure 4.8, which does not show the last row of holes on the suction surface. The cooling scheme at the inner endwall is given in Figure 4.9 shown with circles and elliptic shapes. The crosses indicate the location of the Kapton heat-flux gauges. The temperature measurements from either upper or lower gauge were available at all locations shown. The red rectangles indicate the ones that provided both of these temperature measurements, which resulted in heat-flux measurements. The high-pressure turbine vane is fully cooled, but the down stream rotor blades are un-cooled. The blades do see the cooling flow coming from the vane, and the purge flow coming through the purge cavity.

Figure 4.8 Cooling Hole Geometry on the Vane Airfoil Surface (Not to Scale)
4.3 Instrumentation

There are two total temperature rakes and one total pressure rake located at the inlet and exit of the stage. Static pressure measurements are also obtained for both locations. Each upstream and downstream temperature rake has two RTDs at the base and the upstream pressure rake has one. In addition, the upstream temperature rake has nine miniature butt-welded thermocouples, while the downstream rake has five, which are spaced to cover equal portions of the turbine annulus area. Upstream and downstream pressure rakes consist of five sensors each. Mass flows into and out of the turbine are calculated using the measured inlet total pressure and total temperature. The location of the upstream total temperature rakes were changed circumferentially for some runs in order to observe the circumferential temperature distribution and to determine if the struts have any effect on the overall temperature distribution.

In addition to rake measurements, the inner and outer coolant cavity temperatures and pressures are all measured using miniature butt-welded thermocouples and Kulite
XCQ-062-100A pressure transducers. Vane inner and outer coolant mass flows are calculated using these measurements in conjunction with the choke areas as discussed before. The measurements are obtained on the vane airfoil surfaces, inner endwall and inside the vane airfoils, within the purge cavity, in the angel wing area and for the blade platform, as well as for the blade surface, and the stationary shroud. The outer endwall has cooling holes, but that portion of the vane row was not instrumented.

The vane angel wing cavity area is instrumented with single-sided Pyrex heat-flux gauges (Figure 4.10), miniature thermocouples, and Kulite pressure transducers. The rotor disk and the blades are heavily instrumented with the Pyrex gauges on the surfaces, and with the miniature thermocouples at the leading edge and on the platform.

![Figure 4.10 Pyrex Heat-Flux Gauges in Vane Angel Wing Cavity Area](image)

The vane is instrumented with double-sided Kapton heat-flux gauges at many locations at the inner endwall, and at several span locations of the airfoils for both the pressure and suction surfaces. For this build, eight of the thirty-eight high-pressure turbine vane airfoils were instrumented. There are RTD’s located inside all the instrumented vane airfoils approximately around mid-span to give an additional measure of the wall temperatures. Using the two internal thin film heaters placed inside these vane
airfoils, the wall temperature was varied during some portion of the experiments to obtain adiabatic wall temperature measurements. This will be explained in Section 4.4.

Kapton gauge measurements were obtained at 100 locations, and at 55 of these locations both the upper and the lower gauge were active and thus resulted in a local heat-flux measurement. Table 4.2 shows the distribution of these paired gauges on the airfoil surface. The suction surface of Airfoil 17 has 4 pairs of gauges at the endwall and 1 pair at mid-span. The number of the gauges at each location varies with run due to deterioration of the gauges with time.

<table>
<thead>
<tr>
<th>Vane #</th>
<th>PS # Gauges</th>
<th>Location</th>
<th>SS # Gauges</th>
<th>Location</th>
<th>Alignment</th>
</tr>
</thead>
<tbody>
<tr>
<td>12</td>
<td>6</td>
<td>endwall</td>
<td>2</td>
<td>endwall</td>
<td>MP</td>
</tr>
<tr>
<td>13</td>
<td>4</td>
<td>endwall</td>
<td>4</td>
<td>endwall</td>
<td>VLE</td>
</tr>
<tr>
<td>16</td>
<td>4</td>
<td>endwall</td>
<td>5</td>
<td>50%</td>
<td>MP</td>
</tr>
<tr>
<td>17</td>
<td>3</td>
<td>50%</td>
<td>4+1</td>
<td>endwall+50%</td>
<td>VLE</td>
</tr>
<tr>
<td>31</td>
<td>5</td>
<td>5%</td>
<td>3</td>
<td>5%</td>
<td>VLE</td>
</tr>
<tr>
<td>32</td>
<td>2</td>
<td>15%</td>
<td>3</td>
<td>15%</td>
<td>MP</td>
</tr>
<tr>
<td>35</td>
<td>-</td>
<td>50%</td>
<td>5</td>
<td>50%</td>
<td>VLE</td>
</tr>
<tr>
<td>36</td>
<td>1</td>
<td>90%</td>
<td>3</td>
<td>90%</td>
<td>MP</td>
</tr>
</tbody>
</table>

Table 4.2 Gauge Locations Used For Measurements
A full distribution of all the gauge locations that provided measurements for the airfoil surface is shown on a single airfoil layout in Figure 4.11. The blue circles show all measurement locations, regardless of gauges being in pairs or not. These gauges were either the upper or lower gauges, or both. Therefore, there was always some temperature measurement available from all these gauges. Heat-flux measurements are available for those gauges that have both upper and lower gauges active. Those measurements are shown with red circles. Figure 4.9 showed all gauges providing data at the endwall with crosses. The cooling scheme is indicated with the circular and elliptic shapes. There are roughly two lines of gauges located at the front of the leading edge line, and another line of gauges located at the endwall exit. Figure 4.12 show some heat-flux gauges located at different spans and at the endwall spread out on different airfoils.
Figure 4.12 Heat-Flux Gauges at 90% and 50% Spans (Top), 15% and 5% Spans (Middle), at the Endwall (Bottom)
Figure 4.13 shows the vane row from the forward looking aft, with the relative location of the instrumented airfoils so noted. The front visible section of each airfoil is the pressure surface, and the line separating these sections is the leading edge of each airfoil. For the experimental cases using hot streaks, the set of rods designated with purple circles shows the alignment of the hot streak with the leading edge of every other airfoil, and the other set designated with red circles shows the alignment with the mid-passage of every other airfoil.
The 5%, 15% and 90% span instrumentation are located on one single airfoil per each span. However, due to the higher number of the gauges, the 50% span and the endwall instrumentation have been distributed over multiple airfoils. With this setup, the instrumentation located at different portions of the airfoils will be subject to different hot streak alignment effects, which was summarized in Table 4.2. The mid passage alignment is classified as MP, and the vane leading edge alignment as VLE. The airfoils with pressure surfaces exposed to the hot streak alignment are designated with MP.

![Cross-section of Kapton Gauges](image)

**Figure 4.14 Cross-Section of Kapton Gauges [112]**

It was previously noted that double-sided Kapton heat-flux gauges are used on the cooled vane. This gauge consists of a polyimide (Kapton) insulating substrate and two thin-film nickel resistance thermometers bonded on opposite sides of this substrate, as shown in Figure 4.14. These gauges are originally developed at MIT, and the basic concept has undergone many cycles at The Ohio State University to improve the reliability of the devices. The particular Kapton gauges used for this work were specifically designed at GTL with a modification in the way the connectors from each
surface are mounted, making this newer design structurally much stronger. They are capable of measuring both steady (time-averaged) and unsteady (time-resolved or time-accurate) heat transfer. The upper and lower gauges together give a measurement of heat flux, while still having a very thin structure, being on the order of 0.0011 inches of a thickness. Because the heat flux is deduced using a one-dimensional approximation, it is important to avoid the gauge-to-gauge variation by insuring the precise alignment of the gauges on both sides of the Kapton film. The lower gauge measures the wall temperature, as it is in thermal contact with the surface, while the upper gauge captures the external flow field dynamics.

There are two types of thin-film gauges used for heat-transfer measurements as part of the Build 2 experiments. One of those types mentioned earlier is a single-sided platinum film on a Pyrex substrate, which is the heat-flux gauge originally developed at the Cornell Aeronautical Laboratory in the late 1950s. The disk and blades of the turbine stage is instrumented with these single-sided gauges.

When performing film-cooled experiments, it is necessary to have a local measurement of the surface temperature at the same time as one obtains the external temperature. As a result of the internal flow passages consistent with a cooled component, the local wall temperature has been found to be extremely variable, thus leading to the development of the double-sided heat-flux gauge.

Both the single-sided and the double-sided heat-flux gauges used for the Build 2 measurement program are capable of providing time-averaged and time-accurate heat-flux measurements. They have a high frequency response on the order of hundreds of kHz, which makes detailed unsteady measurements possible. Given the number of the vane and rotor airfoils as 38 and 72 in this series of experiments, the wake and blade passing frequencies in the turbine stage are around 5 and 10 kHz, respectively, at a disk speed on the order of 8000 rpm. In order to resolve high frequency flow data [41], at least twice the highest frequency present in the signal is required as the sampling rate, which the current instrumentation can handle easily. In addition, to be able to resolve the whole frequency range including the harmonic frequencies of primary interest requires at least ten times the highest frequency as the sampling rate. Therefore, increasing the frequency
response level of instrumentation and data acquisition systems enables one to capture more details of the flow field. Of the instrumentation used for Build 2, only the thermocouples have a lower frequency response level, therefore they are used for time-averaged measurements only, as they cannot capture the wake-cutting phenomenon in detail.

4.4 Test Procedure

Operation of the shock tunnel facility in blowdown mode was described in detail above. As a summary, the mass flow through the stage and the pressure ratio across the turbine are controlled via an exit choke, and the vane inlet operating condition (the flow function is duplicated). The rotor torque, the inlet and exit total temperatures and total pressures, and the coolant conditions are measured. The measured coolant and the wall temperatures provide the temperature ratios, \(T_\infty/T_w\) and \(T_\infty/T_c\), for various locations within the stage.

A series of experiments is performed for different engine conditions. For each experiment, the design corrected speed \(N_{\text{corr}}\), flow function (FF), stage pressure ratio (PR), and temperature ratios of gas to wall \(T_\infty/T_w\), and gas to coolant \(T_\infty/T_c\) are matched.

\[
N_{\text{corr}} = \frac{N_{\text{phys}}}{\sqrt{\frac{T_0,\text{inlet}}{T_{\text{ref}}}}} \quad (20),
\]

where \(N_{\text{phys}}\) is the physical speed of the turbine. \(N_{\text{corr}}\) is in units of rpm. \(T_{\text{ref}}\) is a reference temperature of 288K. As an alternative to equation (20), there is also another definition used in industry:

\[
N_{\text{corr}} = \frac{N_{\text{phys}}}{\sqrt{T_0,\text{inlet}}} \quad (21)
\]
In this thesis, equation (21) will be used for corrected speed. With this definition, \( N_{\text{corr}} \) has the units of rpm/\( \sqrt{T} \).

\[
FF = \frac{\dot{m}}{P_{0,\text{inlet}}} \left( \frac{T_{0,\text{inlet}}}{P_{0,\text{inlet}}} \right)^{1/2} \frac{T_{\text{ref}}}{P_{\text{ref}}} \]  
\[
PR = \frac{P_{0,\text{inlet}}}{P_{0,\text{outlet}}} 
\]

(22)  

\( P_{\text{ref}} \) is a reference pressure of 14.7 psia.

The subscripts ‘inlet’ and ‘outlet’ refer to rake locations in above equations (20-23).

As previously noted, there are three different high-pressure turbine vane inlet temperature profiles established by the combustor emulator: uniform profile, radial profile, and hot streak profile, which allow for a comparison of profile effects on heat transfer. Blowing ratio through the cooling holes is varied via different coolant mass flow rates supplied to the vane inner and outer cavities. In addition to the fully-cooled cases, some of the cases are run with no outer vane cooling, no purge cooling, or no cooling at all (one must be careful interpreting these data and the details are discussed in Chapter 5 and Chapter 6).

In order to calculate film effectiveness, one needs adiabatic wall temperature. Because film effectiveness is an important measure for film-cooled experiments, some of the experiments are performed with walls being heated so that this parameter can be calculated. All of the instrumented vane airfoils have thin-film heaters located inside them, allowing for different wall temperatures. Two different wall temperatures and their corresponding calculated heat-flux values give the adiabatic wall temperature (zero heat flux), utilizing a linear relationship between wall temperature and heat flux, as shown in Figure 4.15. This is an approximate method of obtaining adiabatic wall temperatures rather than by directly heating the walls in order to establish a zero heat flux. The latter
technique is more time consuming to accomplish, and it has been demonstrated [9] that the method just outlined works fine.

The temperature criterion is having $\frac{T_\infty}{T_c} = 2$, and therefore, this parameter is matched during the experiments. $\frac{T_\infty}{T_w}$ was automatically satisfied without any additional efforts. Heating the walls to establish the adiabatic wall conditions did not alter $\frac{T_\infty}{T_w}$ noticeably. The other parameter of interest is the Net Heat Flux Reduction, as defined in equation (4), or Net Stanton Reduction, as defined in equation (10), which are the measures of overall performance of film cooling process. In order to calculate this parameter, no cooling cases are needed.

4.5 Data Collection

A measurement program of the type under discussion here requires a very large number of high sampling frequency data acquisition channels. The data from the heat-flux gauges and pressure transducers is collected using two main data acquisition systems (DAS): A 256 channel, 100 KHz/channel sampling frequency, 12-bit resolution, CAMAC based system, and a 388 channel, maximum of 2.5 MHz/channel (for these experiments sampling was done at 250 KHz/channel), 16-bit resolution, VXI based system, which uses 8 channel VX2824B modules. This large range of system acquisition
rates allows for the detailed time-accurate measurements. Additionally, two standard National Instruments PCI-6071 board based units, one during pre-experiment and one during the main experiment are used to sample the rakes, the combustor emulator, the cooling tubes, and LCF instrumentation. The DAS systems use Butterworth Anti-Aliasing filters at around half of the sampling rate.

The raw data collected during the runs are recorded by these systems in terms of volts. A 1 mA current is applied to the heat-flux gauges and the voltage output is measured as a function of time during the experiments. This gives the time-variation of resistance of each gauge, and with the incorporation of calibration data, gauge surface temperatures are calculated for both sides of the Kapton gauges. A photograph of the digital data acquisition system is given in Figure 4.16.

In order to understand how the measurements reported in this thesis are performed, it is instructive to look at typical upstream and downstream total pressure and
total-to-total pressure ratio measurements as a function of time. Figure 4.17 and Figure 4.18 present typical time histories of the parameters just noted. Time equal to zero on these plots is the time at which the digital data acquisition system (DDAS) is triggered. However, at the time of the DDAS trigger the rotor has already been brought to a stable speed, which is a value just below the anticipated physical design speed appropriate for the soon-to-be-realized flow conditions. The rapid increase in pressure shown on the upstream pressure measurement of Figure 4.17 at about 550 ms is indicative of the arrival of the mainstream flow from the supply tube. For the experiments using coolant gas, the coolant gas supply is activated approximately 250 ms prior to the mainstream flow. The time interval between about 550 ms and 630 ms is indicative of the stage flow establishment process during which the turbine design flow function and stage pressure ratio are established. The flow establishment process consists of a series of acoustic waves communicating between the exit nozzle and the inlet to the rig housing the turbine stage. This flow establishment process occurs in a relatively short period of time after which the mass flow rate through the turbine and various parameters are constant (steady state) as illustrated by the stage pressure ratio shown in Figure 4.17. It is at this time that the target point is achieved, with the matched corrected speed, flow function, and pressure ratio. At a time in the vicinity of 700 ms, the FAV is closed, terminating the airflow to the dump tank, and therefore the pressure levels decrease.
Figure 4.17 Upstream and Downstream Total Pressure Measurements From the Rakes

Figure 4.18 Pressure Ratio Variation During Operation
Figure 4.19 Turbine Speed Variation During Operation

Figure 4.19 is a time history of the turbine physical speed during the data-recording period. As was previously mentioned, the disk is initially brought up to a speed in the evacuated dump tank to a value that is on the order of 1% below target point. Once the main tunnel valve is opened and air begins to flow through the turbine stage, the turbine extracts work from the flow and speed increases rapidly. As soon as the FAV is closed, the disk starts decelerating (which reflects as a slight decay on the plot after 700 ms), and then an external valve is opened to pressurize the dump tank and to help to slow the disk. Generally, it takes about 10 minutes for the disk to come to a full stop.

The vane heat-flux gauge instrumentation can also be used to demonstrate the pattern of the flow establishment process as demonstrated on Figure 4.20. This figure presents a typical temperature-time history measurement obtained from the lower side of a Kapton gauge. From time equal to zero to about 300 ms the temperature is constant, but then changes with the introduction of the coolant gas at around 300 ms. When the tunnel FAV is opened, and the airflow starts at around 550 ms, an immediate sharp rise occurs. The compression heating associated with the starting process can be seen by the peak temperature recorded in the vicinity of 625 ms, which is followed by the uniform temperature during the test time period shown in the vicinity of 700 ms. The pressurized
air is introduced into the turbine rig for about 125 ms duration. Finally, the FAV is closed around 700 ms, which is reflected as a temperature drop. This entire process results in an overall data collection period of nearly 1-s for the digital data acquisition system. Of this period, the region of interest is the target point, which only lasts around 14 ms for a 2-revolution of rotor design speed. The inlet bulk temperature levels achieved during the experiments are in the range of 500-600 K. The walls remain at a near isothermal condition since there is not enough time for significant heat transfer.

Figure 4.20 A Typical Temperature Measurement From the Lower Side of a Kapton Gauge

When the pressure and the temperature plots are compared, the initial drop in the pressure peak during the flow establishment period takes place earlier and at a shorter time than the drop in temperature peak. Even if the frequency response of the Kapton gauge is high, because there is a thermal boundary layer being established along the wall the temperature peak follows the pressure peak with a time delay.
Figure 4.21 A Typical Temperature Measurement From Both Sides of a Kapton Gauge

Figure 4.21 shows measurements from both upper and lower sides of a typical double-sided heat-flux gauge. The upper measurement clearly shows local activity due to exposure to the turbine flow path, while the lower measurement shows much reduced activity. The lower heat-flux gauge is a measure of the airfoil surface temperature and both gauges reflect the arrival of cooling gas inside and outside the airfoil that was not present at a time prior to about 300 ms.

The time range of measurements is presented in the previous figures, with the target point designated with bars on the corresponding portion of the measurement. The results of this thesis will mainly concentrate on heat-flux measurements at different airfoil and endwall locations. Heat flux is obtained using the temperature measurements obtained from upper and lower sides of a Kapton heat-flux gauge. The data that is soon to be presented reflects the time-averaged values of these measurements at the target point. The time windows of the vane measurements are about 10 ms earlier or later (depending on the run) on average than the windows used for the data analysis of the downstream rotor in order to capture a more stabilized temperature measurement region. Because the cooling level variations affect the pressure ratio, the pressure ratio does not exactly match
the design value. As a result, the time window is defined primarily depending on the isothermal region of the upstream rake measurements, and the pressure ratio is allowed to slightly deviate from the design value. This slight change in the target point did not alter the experimental operating conditions.

4.6 Calibration and Data Reduction

Pressure transducers are calibrated throughout the matrix by pressurizing followed by depressurizing the dump tank in which the test rig is located while monitoring the output signal through the entire DAS. Whether they are total pressures mount on the rake or static pressures mounted in the cooling cavities, the main calibrations come from comparing these sensors to a standard mounted in the dump tank, a Heise HPO 150-psia pressure sensor, which has a rated accuracy of 0.05%. Each run is checked for offset changes by comparing to the standard. Past calibration of total pressure sensors in the calibration rig have shown that no correction is needed for the pressure sensors installed in a “total pressure” configuration.

The thermocouples located at both the inlet and the exit rakes are calibrated in an isentropic blowdown rig, and the heat-flux gauges on individual rotor airfoils are calibrated using an oil bath. Because the vane airfoils are welded together in a large ring, it was not possible to disassemble the vane row to perform individual vane calibrations in the oil bath, so the vane instrumentation was calibrated as a unit in the dump tank of the TTF facility. The vane was put in contact with a ring that was directly heated to high temperatures and allowed to cool down slowly. This ring was surrounded by a plexiglass container, which was further wrapped with bubble wrap to avoid radiation effects. The surface temperature measurements necessary for the calibration were obtained using RTD’s located all around the vane surface. Since the vane row is insulated during this process, the only mechanism causing heat transfer is conduction through the vane. The pace of the calibration process is designed to be sufficiently slow to establish a quasi-static system. The calibration temperature values are chosen to mimic the peak values during the experiment in order to ensure measurement accuracy during the measurement program.
4.6.1 Calibration of Kapton Heat-Flux Gauges

The Kapton heat-flux sensors relate measured resistance to temperature. The temperature time histories differences between the upper and lower sensor temperatures provide the heat-flux. For all instruments, the raw data is acquired in terms of volts, and must be converted to engineering units. For the heat-flux sensors, this means that natural resistance measurements are converted to voltage using a preamplifier, which provides a known current, an offset, and a preamplifier gain to the signal. These values are a function of the amplifier only, and can be measured at any point in time. However, if the measurements are made at a point where the temperature of the model is well known, then the actual operation of the sensor can also be determined. This is usually done before any of the heaters are turned on and form a baseline at a known laboratory temperature (T_{lab}), and the amplification setting (preamp gain (G)), Voffset, and the current (I) are recorded and stored. One has to also account for the line resistance (the effect of the resistance from the gauge to the amplifier). This is generally done at the same time (since the rig temperature is well known), if the calibration coefficients that relate the temperature to the resistance are known from a laboratory based calibration. However, given the very long test matrix, a calibration technique was developed that used data available during the run matrix to produce improved scale factor calibrations. These were in general quite close to the original, but any gauges that suffered some damage that changed their resistances were still able to be used using this technique, as the offset term and the line resistance no longer mattered. Thus, the scale factor was calculated using a linear regression technique using measured resistances and temperatures as indicated for each run by the RTD’s mounted inside eight instrumented vanes. This technique focused on changes in temperature and changes in resistance, and thus did not care about offsets. Different regression techniques were used which had different sensitivities to outlying data, but the variation due to processing was very small. The final temperature is calculated as:

\[ T_{\text{experiment}} = T_{\text{ref}} + a_1 \Delta R_{\text{experiment}} \]  

(24)
where $T_{\text{ref}}$ is the initial temperature from the RTD at the beginning of the test, and $a_1$ is the scale. $\Delta R_{\text{experiment}}$ is the change in resistance from the beginning of the test. Once the temperature is produced, a time-window is selected which allows the experimental operating conditions to be matched.

This time window is taken as two revolutions (2-rev) of the disk, which maintains the corrected speed within a $\pm 1\%$ variation. Two revolutions of the rotor give plenty of data to analyze the unsteady interaction of the vane row with the blade row. The criterion is to select the time window of interest in such a way that the experimental condition matches the design flow function, design stage pressure ratio, and design corrected speed. At the same time, the measured inlet bulk temperatures at the location of the inlet total rakes during that time interval should be relatively constant. In order to take accurate measurements, this 2-rev time window needs to be stable. The influence of blade passing on the various vane measurement locations is verified by performing a FFT (Fast-Fourier Transform) analysis on the measured parameter, which will be demonstrated in the data analysis.

4.6.2 Data Processing

The intent of the data system used for this work is to record as much of the frequency content of the data as possible with a minimum of data filtering. If one has foreign noise in the data file, it can always be filtered out at a later date. However, if one fails to record the entire frequency content of the data stream, those data can never be recovered. The data system used in this work has been previously described in several OSU GTL published papers.
In some cases, Running Average and Thinning processes are applied to the data that was specifically collected by the VXI system. Because this system has a higher acquisition rate than the CAMAC system, the data it collects is sampled at a higher rate. Applying a 10-point running average technique helps to reduce the time variation from the signal. The temperature signal is not significantly affected with these steps, but the heat-flux signal is considerably cleaned up. The latter step is to apply a sub-sampling technique (Thinning) that takes only every 5th sample into records. These two processes therefore were not applied on the raw data collected by the CAMAC system. Figure 4.22 shows a sample measurement before and after processing is applied. With filtering, the noise spikes appearing in the raw data are removed. However, the time-averaged value at target point has varied only by 0.001% of the raw measurement for the gauge shown. In an average, there is about a 0.02% reduction in the time-averaged temperature measurement from the raw measurement with the application of processing. This variation adds up to only 0.05% of the raw measurement for the heat flux values. Thus the variation caused by processing of the raw measurement is very small, and can be ignored.
4.6.3 ACQ

The code used to obtain heat flux from the raw data for the purposes of this research effort is the code referred to as ACQ that has its origin in the ACQ code that was originally developed at MIT in 1984 for the United States Air Force for the data reduction from the double-sided heat-flux gauges being developed at MIT. The main algorithm currently used at the OSU GTL was adopted from the original ACQ mentioned above, but it was further improved at the GTL as mentioned in the technical report of Weaver et al. [112]. The single-sided gauge data reduction for the rotor is performed using Cook-Felderman algorithm, which was perfected at Calspan. Cook-Felderman algorithm is also used as a check on ACQ during its development at the GTL. Hence, ACQ is applicable to both single-sided and double-sided gauges.

The unsteady one-dimensional heat conduction equation (25) is solved numerically, with the boundary conditions obtained from the upper and lower gauge surface temperatures. Then, the heat-flux time history through the substrate is calculated via (26).

\[
\frac{\partial T}{\partial t} = \frac{k}{\rho C_p} \frac{\partial^2 T}{\partial x^2} \quad (25)
\]

\[
q'' = -k \frac{\partial T}{\partial x} \quad (26)
\]

The numerical solution procedure allows for a time varying temperature history making it possible to deduce heat flux for any portion of the time period after the experiment start time. To derive the non-uniform initial condition, it is assumed that there is a linear temperature distribution through the substrate at the start time.

\[
T(x) = T_u(0) + (T_l(0) - T_u(0)) \frac{x}{d} \quad (27),
\]

where \(T_u(0)\) and \(T_l(0)\) are the upper and lower gauge temperatures at \(t=0\), respectively.
Here, x is the depth into the substrate measured from the upper surface. The algorithm splits the 0.001-inch Kapton thickness into many layers, and performs these calculations on each layer separately, carrying along the result of one layer to the next. For further information, the reader is referred to the technical report [112] mentioned above that describes the modifications done to the original ACQ approach and describes the data analysis process in depth.

4.6.4 Fast-Fourier Transform (FFT)

The Fast Fourier Transform, FFT, analysis is a very useful tool to determine the primary and secondary effects causing the unsteadiness in the flow environment where highly unsteady interactions as in this research are experienced. It enables one to use the time history of a given signal and present the information contained in that signal in the frequency domain. By presenting the data in the frequency domain as power spectrums, the primary (fundamental) frequency as well as the other important ones (harmonics) in the flow field can be analyzed. The algorithm uses the theory of Fourier series, which is that every periodic function can be written as an infinite sum of sinusoidal functions. It gives the combination of the amplitudes and the phase of different sinusoidal waves which the signal consists of at any given time window over the experiment duration.

For the specific case of wake-blade interaction, the unsteady content of this flow phenomenon is observed as a set of frequencies. The fundamental frequency represents the blade passing frequency if the measurement of interest is for the vane frame of reference, or the wake cutting frequency if it is for the blade frame of reference. The remaining harmonics show the energy content in the remainder of the frequency domain. Dunn et al. [113] provide a detailed description of FFT analysis on time-accurate measurements. Temperature history consists of fluctuations due to both wake cutting and the existence of turbulence and secondary flows in the NGV flow field. In cases where the vane exit Mach number is high enough, shock waves will also contribute to the unsteadiness, such as in a transonic turbine used for this research. The suction surface trailing edge region and the endwall exit are the locations that observe the disturbances caused by the downstream rotor. Using the FFT approach, the peaks at the fundamental and first harmonic frequencies can be resolved. Hence, the unsteady (time-accurate)
parameters such as temperature and pressure can be presented in a frequency domain, and the extent of unsteadiness can be inferred from the amplitude variation.

As an example, an FFT analysis on the mid-span gauges for the un-cooled run is demonstrated in Figure 4.23, spanning different wetted distance locations on the airfoil. Colors show measurements coming from the gauges located at various distances at the mid-span on both pressure and suction surfaces. The suction surface of the airfoil at the aft of the throat that is around 40% wetted distance is the region where the downstream effects can be observed the most. Since the throat is choked, the downstream flow effects cannot be transferred upstream of the throat area. Hence, the effect of rotation will be confined to the trailing region on the suction surface.

![Figure 4.23 Unsteady Effects Due to Downstream Rotor Rotation Captured by 50% Span Gauges](image)

As a consequence, there is not much action on the pressure surface. The very low level of amplitudes for the two different wetted distance locations designated as “WD” show that this surface of the vane airfoil is not affected by the downstream rotor. On the other hand, each blade passing of the downstream rotor is causing a high level of unsteadiness on the suction surface of the vane airfoil. This is observable for the last two gauges that are beyond the throat location via significantly high amplitude of
fundamental frequency. Blade passing frequency is on the order of 10 kHz, and is followed by other harmonics showing the energy content in the flow field. The blade passing frequency is the most observable one by the last gauge that is at 86.1% wetted distance. Since the throat of the airfoil is roughly located around 40% along the wetted distance, the first gauge at 17.5% wetted distance on the suction surface closer to the leading edge does not observe blade passing.

![Figure 4.24 Effects of Downstream Rotor Rotation at the Endwall Exit](image)

Figure 4.24 shows the contribution of the rotation of the rotor to the unsteadiness of the flow field via a comparison of the non-rotating and rotating runs at the endwall exit. The cold runs do not have a temperature profile imposed at the inlet. The radial run is included in the comparison to observe the effect of the heated environment on the rotation. For the stationary rotor, there is no frequency content in the flow field as there are no energizing sources. Rotation of the rotor introduces some action (unsteadiness) to the flow field, causing a significant rise in the energy content. On top of that, introducing a radial temperature profile and elevating the temperature levels make the effect of rotation more significant. In the heated environment, the fundamental frequency amplitude is doubled.
Unsteady measurements are generally demonstrated either in the frequency or the time domain. When using the frequency domain to present the results, they are shown as power spectrums obtained by FFT technique, as shown above. When presenting the results in the time domain, the results do not contain any phase information, while the FFT presentation does. It is very common to present the data using the FFT when one is interested in aeromechanic issues. On the other hand, for the time domain it is common to use ensemble plots that are phase-locked to the blade passing time scale, since this is one of the phenomena of interest to the aerodynamics community. For a specified location, the variation of the data within the range of the unsteady envelope provides a measure of the maximum and minimum points of the variation. It is important to examine the measurements in both the time domain and frequency domain.

Although all measurements obtained in this experimental program get their share from the unsteadiness caused by the effects such as the rotation of the downstream rotor and the shock waves, for the data presented in this thesis, the focus is not on the unsteady content of the measurements. The data analysis will consist of the time-average of the measurements instead.

The goal of this chapter was to introduce the experimental setup, the test procedure, and typical flow parameters measured via the use of data acquisition systems. The primary results obtained for the cooled vane row are presented in the next chapter.
CHAPTER 5
TURBINE STAGE OPERATING CONDITIONS USED FOR MEASUREMENT PROGRAM AND DATA QUALITY

A major goal of this experimental effort has been to quantify the effects of two parameters that influence vane heat transfer: inlet temperature profiles, and cooling flow amounts (the inlet conditions). It is important to establish a good understanding of the inlet conditions in order to interpret the data obtained throughout the vane section hardware. This chapter shows that the high repeatability of these inlet conditions provide confidence that the variations in the measured heat flux among the experimental conditions shown in Chapter 6 are due to the changes in inlet conditions, and not due to the uncertainty in these inlet conditions. The discussion starts with the turbine stage experimental operating conditions and the overall characterization of the temperature and pressure profiles at the inlet to the vane. These profiles form the basis for the control variables used in the analysis. A rather detailed discussion of measurement quality and uncertainty follows in order to quantify these variations.

5.1 Experimental Matrix and Control Variables

A wide range of experimental operating conditions resulted in an experimental matrix for Build 2 consisting of 50 runs at two distinct pressure ratios. Of these, 35 runs represent a range of cooling conditions, temperature profiles, and repeat experiments (Table 5.1) all at one set of pressure ratios. Runs are classified according to the temperature profiles at the high-pressure vane inlet, and the cooling levels supplied to different parts of the turbine stage. Each temperature profile is highlighted by a different color, with repeat runs listed in the same row and the alignment and magnitude status for the hot streaks. The instrumented vanes were equipped with internal heaters to change the metal temperature between runs to determine the adiabatic wall temperatures.
<table>
<thead>
<tr>
<th>RUN#</th>
<th>INLET CONDITION</th>
<th>COOLING AMOUNT</th>
<th>COOLING LEVEL DEFINITION</th>
<th>HOT STREAK DEFINITION</th>
<th>ADIABATIC WALL HEATERS</th>
<th>DOWNSWEEPER ROTOR STATUS</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Temperature Profile</td>
<td>Vane Inner</td>
<td>Vane Outer</td>
<td>Purge</td>
<td>Alignment With</td>
<td>Magnitude</td>
</tr>
<tr>
<td>15,16</td>
<td>Cold</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>20,21</td>
<td>Cold</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>43,47</td>
<td>Uniform</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>Un-cooled</td>
<td>--</td>
</tr>
<tr>
<td>31</td>
<td>Uniform</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Nominal</td>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>32</td>
<td>Uniform</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Nominal</td>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>33</td>
<td>Uniform</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Nominal</td>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>49</td>
<td>Uniform</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Nominal</td>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>22</td>
<td>Radial</td>
<td>Low</td>
<td>Low</td>
<td>High</td>
<td>Low</td>
<td>--</td>
</tr>
<tr>
<td>17,23,24</td>
<td>Radial</td>
<td>Low</td>
<td>Low</td>
<td>High</td>
<td>Low</td>
<td>--</td>
</tr>
<tr>
<td>18,19</td>
<td>Radial</td>
<td>Low</td>
<td>Low</td>
<td>--</td>
<td>Low</td>
<td>--</td>
</tr>
<tr>
<td>25</td>
<td>Radial</td>
<td>Low</td>
<td>Low</td>
<td>Nominal/Low</td>
<td>Low</td>
<td>--</td>
</tr>
<tr>
<td>40,41</td>
<td>Radial</td>
<td>Nominal</td>
<td>--</td>
<td>Nominal</td>
<td>Nominal-No Vane Outer</td>
<td>--</td>
</tr>
<tr>
<td>26</td>
<td>Radial</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Nominal</td>
<td>--</td>
</tr>
<tr>
<td>27,28</td>
<td>Radial</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Nominal</td>
<td>--</td>
</tr>
<tr>
<td>44,45</td>
<td>Radial</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Nominal</td>
<td>--</td>
</tr>
<tr>
<td>50</td>
<td>Radial</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Nominal</td>
<td>--</td>
</tr>
<tr>
<td>29,30</td>
<td>Radial</td>
<td>High</td>
<td>High</td>
<td>Nominal/High</td>
<td>High</td>
<td>--</td>
</tr>
<tr>
<td>46</td>
<td>Hot Streaks (HST)</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>Un-cooled</td>
<td>--</td>
</tr>
<tr>
<td>42</td>
<td>Hot Streaks (HST)</td>
<td>Nominal</td>
<td>--</td>
<td>Nominal</td>
<td>Nominal-No Vane Outer</td>
<td>Mid-Passage</td>
</tr>
<tr>
<td>34</td>
<td>Hot Streaks (HST)</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Mid-Passage</td>
</tr>
<tr>
<td>35</td>
<td>Hot Streaks (HST)</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Mid-Passage</td>
</tr>
<tr>
<td>36</td>
<td>Hot Streaks (HST)</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Mid-Passage</td>
</tr>
<tr>
<td>37</td>
<td>Hot Streaks (HST)</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Vane Leading Edge</td>
</tr>
<tr>
<td>38</td>
<td>Hot Streaks (HST)</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Vane Leading Edge</td>
</tr>
<tr>
<td>39</td>
<td>Hot Streaks (HST)</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Vane Leading Edge</td>
</tr>
<tr>
<td>48</td>
<td>Hot Streaks (HST)</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Nominal</td>
<td>Mid-Passage</td>
</tr>
</tbody>
</table>

Table 5.1 Experimental Matrix
In Table 5.2, the runs are grouped with respect to their cooling level status regardless of the temperature profile types. The cold runs (15, 16, 20) did not have a temperature profile. Run 21 had a very low temperature uniform profile and is considered a cold run. These runs are not included in Table 5.2. The pressure ratio and the corrected speed for each cooling level are provided along with their target design values, other than the flow function. The gas-to-coolant temperature ratio \(\frac{T_\infty}{T_c}\) was kept at 2 during the experiments.

The variations in all parameters are defined as the half of the range divided by the average value. The variations for the ratios are defined as the range (maximum-minimum). For example, for the runs with low cooling level, the ratio of the inlet temperatures to the average of temperatures for all runs mentioned in the table, \(T_{\text{avg}}\), varies from 0.99 to 1.02. This gives a 0.002\% variation around the average ratio for low cooling level when calculated as half of the range divided by this average value. Similarly, the vane inner coolant/total ratio varies from 4.49\% to 4.76\%. This gives an error of +0.12 and –0.15. The maximum absolute of these two values is used to define the overall variation, resulting in ±0.15. This definition of repeatability helps to provide a measured value (as opposed to a calculated one such as the standard deviation) for the general repeatability of the average values. As will be demonstrated in Figure 5.4, where a more detailed statistical analysis has been done, and 95\% confidence limits established, this range as a measure of stability is a good approximation when dealing with a small number of samples.

The cooling split within the circuits is also provided in Table 5.2. For all cooling cases, the vane inner coolant is supplying coolant gas to the airfoil surfaces and the inner endwall, and the vane outer coolant is supplying coolant gas to the trailing edge, a few additional rows of holes on the airfoil surface at the trailing edge on the pressure surface, and the outer endwall regions. ‘Total’ refers to the sum of the core flow and the overall amount of coolant, i.e. the vane inner, the vane outer, and the purge flow. Purge flow is the coolant gas supplied to the disk cavity and does not contribute directly to the cooling of the vane surface. However, the interaction of this purge flow with the vane exit flow is shown to be a complicated phenomenon, and any level of purge cooling is found to have
noticeable effects on the blade surface and platform heat transfer despite the relatively smaller mass injected through the purge cavity [83]. The influence of purge flow will also be investigated for the vane heat transfer. The core flow is calculated by using the total pressure and temperature measurements obtained by the upstream rakes, and the effective vane choke area, which changes with cooling level (last row) and is estimated using the un-cooled runs as a basis. This is explained in more detail in the next section.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>P_{0,inlet}/P_{0,outlet}</td>
<td>4.74</td>
<td>4.62±2.67%</td>
<td>4.53±1.74%</td>
<td>4.59±1.31%</td>
<td>4.61± 1.67%</td>
<td>4.56± 0.05%</td>
</tr>
<tr>
<td>P_{0,inlet}/P_{0,outlet}</td>
<td>5.68</td>
<td>5.66± 4.07%</td>
<td>5.49± 2.12%</td>
<td>5.58± 2.13%</td>
<td>5.59± 0.70%</td>
<td>5.56± 0.06%</td>
</tr>
<tr>
<td>T_{0,inlet}/T_{avg}</td>
<td>1.02±0.03%</td>
<td>1.01±0.00%</td>
<td>0.99±0.03%</td>
<td>0.97±0.00%</td>
<td>1.01±0.00%</td>
<td></td>
</tr>
<tr>
<td>m, Inlet Mass Flow Rate, kg/sec</td>
<td>15.00±6.0%</td>
<td>14.52±2.0%</td>
<td>14.22±5.6%</td>
<td>14.68±1.2%</td>
<td>14.02±0.8%</td>
<td></td>
</tr>
<tr>
<td>N_{corr} (rpm/√K)</td>
<td>359.62</td>
<td>373.9±3.0%</td>
<td>364.5± 2.2%</td>
<td>371.9± 2.2%</td>
<td>372.7± 0.7%</td>
<td>369.4± 0.5%</td>
</tr>
<tr>
<td>% of N_{corr}</td>
<td>103.98%</td>
<td>101.35%</td>
<td>103.41%</td>
<td>103.64%</td>
<td>102.72%</td>
<td></td>
</tr>
<tr>
<td>%Vane Inner Coolant/Total</td>
<td>0</td>
<td>4.64%±0.15</td>
<td>6.89%±0.40</td>
<td>7.06%±0.06</td>
<td>8.06%±0.0</td>
<td></td>
</tr>
<tr>
<td>%Vane Outer/Total</td>
<td>0</td>
<td>4.35%±0.14</td>
<td>5.37%±0.31</td>
<td>0</td>
<td>6.28%±0.0</td>
<td></td>
</tr>
<tr>
<td>%Purge/Total</td>
<td>0</td>
<td>0.94%±0.94</td>
<td>0.69%±0.04</td>
<td>0.71%±0.01</td>
<td>0.82%±0.0</td>
<td></td>
</tr>
<tr>
<td>Vane Area (% of Un-cooled)</td>
<td>99.77%±0.04</td>
<td>94.92%±0.02</td>
<td>90.88%±0.03</td>
<td>93.61%±0.01</td>
<td>90.43%±0.01</td>
<td></td>
</tr>
</tbody>
</table>

Table 5.2 Typical Experimental Conditions for Which Data Were Obtained

The change in the effective vane choke area as coolant is added through the cooling circuits is shown in Figure 5.1, but not all cooling circuits impact the choke area. The range bars that show the variation in the calculated throat area and the measured coolant amounts are generally very small. The range bar at around 95% on the x-axis is large, since this is the area ratio corresponding to the low cooling level for which the runs
used have different purge conditions varying from high to none. When there is no cooling, the throat area is at its maximum.

![Figure 5.1 Vane Throat Area Change with Cooling Flow](image)

Purge flow does not have an effect on the throat area, as it is added downstream of the throat area through the wheel space cavity between the vane and the rotor. The vane outer cooling circuit has a partial effect on the choke area. The outer endwall cooling and three of the sixteen airfoil cooling hole rows flow through the choke, but the majority of the outer cooling circuit gas exits out the trailing edge, not affecting the choke area. The total vane inner and vane outer line that is shown with the blue curve follows a smooth trend, which reflects the increase in the effective choke area with a decrease in the coolant amounts supplied.

The coolant flow rates and areas mentioned in Table 5.2 are used to generate this plot with the exception of the no vane outer cooling case. Because for this experimental condition, there is localized ingestion in the endwall, and thus the area being estimated...
includes both the choke area and the effective bypass through the cooling holes. This is a different physical situation than the other cases and is thus not included in the plot.

Since the addition of cooling changes the throat area, the throat area is another parameter introducing variation between different cooling flow conditions. This variation source is removed by the use of Stanton numbers as well.

5.1.1 Mass Flow Balances

The new vane choke areas for each cooling level are calculated in such a way that the mass balance between the inlet and the exit flows of the turbine stage should be satisfied for each group of runs. This is done by finding an average area for the un-cooled cases, which equates the exit choke area of the rig to the vane choke area. Since the exit area does not change for the runs at any given pressure ratio, and the mass flow for the cooling circuits are well known (being measured using a blowdown procedure described in [1-2]), the average choke area for all the runs at a given vane flow cooling flow rate can be estimated providing on average a mass balance. Variations for each individual run from the group represent the accuracy of the mass balance. Currently, the run-to-run variation in the mass flow is less than ±2% of the exit mass flow for the majority of the runs. This is an important check, as the mass flow balance requires accurate estimations of the average inlet and exit temperatures for the calculations, thus making this a good check on the overall temperature profiles.

5.1.2 Inlet Temperature Profiles

The combustor emulator used to create the temperature profiles for this experiment is a passive heat exchanger consisting of a honeycomb matrix located upstream of the turbine stage described in [1-2, 9]. For this experimental series, there were two temperature rakes located upstream of the vane inlet, instrumented with butt-welded miniature thermocouples measuring the total temperature at several different span locations. These rakes could be moved to different circumferential locations (8 possible locations), although this was seldom done in this series of experiments. The two rakes are located at ±45° from either the top or the bottom dead center for the majority of the runs. For Runs 46-49, one of the rakes was moved behind the strut at 90° from the dead center. This provided a check on the circumferential uniformity since the temperature profile
created by the combustor emulator should be circumferentially uniform for the radial and uniform profiles so that different vanes are exposed to the same inlet condition. Considering that the measurements are not confined to only one airfoil, but rather distributed throughout the vane row (Figure 4.13), satisfying this criterion is critical, and the resulting circumferential variation is not significant.

One goal of this chapter is to isolate different effects influencing vane heat transfer, and account/remove the possible sources of the variations reflected in the heat transfer measurements from the control variables. One possible source is the changes in the inlet temperature profile shape and average value. Comparisons for the profiles between runs will be performed via normalizing all temperature measurements by the average temperature of both rakes for that run. However, one should keep in mind that any of the definitions given by equations (17-19) could also be used for this purpose.

The resulting temperature profiles measured by these inlet rakes are shown in Figure 5.2 for each profile shape. The normalized temperature for each run by its average temperature from both rakes ($T_{avg}$) are averaged together to produce the values for each profile shape. The range bars show the peak-to-peak variation (maximum-minimum) among all runs within that profile. $T_{avg}$ here is the inlet bulk temperature, $T_{0,inlet}$, mentioned in Table 5.2.

Figure 5.2 shows the span wise distribution of each type of inlet temperature profile, which illustrates that the differences among the profiles are well pronounced as intended. All runs of the experimental matrix are used in the profile shapes shown. These runs are given in Table 5.3. The uniform profile represents the average of 6 available uniform runs, and it resembles the cold profile very closely (even though the average temperatures are quite different), with a very tight variation of $\pm1.8\%$. There are a total of 17 runs with five different cooling cases for the radial profile, and 9 runs with different alignments and magnitudes for hot streaks. Although the rakes would be ideally lined up exactly with the location of the hot streaks, this was not possible due to the way the heater matrix was constructed. Hence, there is also a variation introduced by the pitch-wise location of the rakes with respect to the hot streaks in addition to the alignment and magnitude effects. These factors all contribute to the larger variations observed for these
profiles, ±3.9% for the radial profile, and ±8.5% for the hot streaks. These variations are calculated using all available runs for each profile, and are not confined to the repeatable runs only. If these experiments were to be reinstituted at a later date, the number of rakes and the thermocouples per rake would both be increased in order to improve the inlet temperature profile resolution.

![Inlet Temperature Profiles Measured at Upstream Rakes](image)

**Figure 5.2 Inlet Temperature Profiles Measured at Upstream Rakes**

<table>
<thead>
<tr>
<th>COOLING LEVEL</th>
<th>UN-COOLED</th>
<th>LOW</th>
<th>NOMINAL</th>
<th>HIGH</th>
</tr>
</thead>
<tbody>
<tr>
<td>RADIAL RUNS</td>
<td>22</td>
<td>[17-19], [23-25]</td>
<td>[26-28], 40,41,44,45, 50</td>
<td>29,30</td>
</tr>
<tr>
<td>PROFILES</td>
<td>COLD</td>
<td>UNIFORM</td>
<td>RADIAL</td>
<td>HOT STREAKS</td>
</tr>
<tr>
<td>RUNS</td>
<td>15,16,20,21</td>
<td>[31-33], 43,47,49</td>
<td>[17-19], [22-30], 40,41,44,45,50</td>
<td>[34-39], 42,46,48</td>
</tr>
</tbody>
</table>

**Table 5.3 All Runs Presented For Temperature Profiles and Cooling Levels**
Figure 5.3 Inlet Temperature Profiles Measured at Upstream Rakes With Different Cooling Levels
Figure 5.3 shows the differences among the measurements from both rakes for all profiles, and the profiles are further split into different cooling levels. Low cooling level includes the ‘low cooling no purge’ case, and the nominal cooling level includes the ‘nominal cooling no vane outer’ case as shown in Table 5.3. The right-hand side column gives the calculated difference between the average values of the two rakes at each span location. The measurement from the thermocouple located at 10% span was not available at one of the rakes, and was duplicated for the other rake. As a result, this data point is not presented in the figures of the right-hand side column. Different cooling levels have very similar profile shapes. That is, the differences between them stay within the variations of each cooling level.

The rake-to-rake variation is generally within ±5% of the average temperature for the radial profiles, and even tighter for the uniform profiles. The group of runs used in each cooling level includes both repeat and not repeat runs whenever available. When the analysis is confined to repeat runs only, the average variation gets even tighter for both radial and uniform profiles, decreasing to ±2% of the average temperature. The only exception is the repeat group for radial profile at high cooling level, with a variation of ±3.5%. None of the hot streak groups shown in Figure 5.3 have repeat runs in them. The difference between the two rakes increases dramatically for the un-cooled case of the hot streaks. The reason for this is the change in the rake locations during the runs. One should note that the locations of the rakes were changed circumferentially for some runs in order to observe the circumferential variation. The un-cooled hot streak run, Run 46, had the second rake located behind one of the struts. This is the main reason why the measurements from this rake differ compared to the rest, and shows that the struts do have an effect on the overall temperature distribution. The other runs that had the second rake behind the strut are Runs 47, 48, 49. Run 47 is an un-cooled case of radial profile, and is the one causing the largest variation also, according to Figure 5.3. Runs 48 (hot streak) and 49 (uniform) are the runs where the walls were heated. This fact dominates over the effect of the strut in circumferential variation, and as a result the strut existence does not contribute as much to the overall circumferential variation. Therefore, except Run 46, the circumferential uniformity is satisfied within ±5% of $T_{avg}$.
Having this variation small is important, since it ensures that each airfoil section in the vane is exposed to the same inlet conditions for the radial and uniform profiles, so that a one-to-one comparison can be made between measurements spread over the vane row. On the other hand, one must ensure that the flow properties at the inlet, other than those that are intentionally varied for testing purposes, remain the same for each run so that a direct comparison between the runs can be made. As can be inferred from Table 5.2, the variation in inlet bulk temperature, $T_{0,\text{inlet}}$ (or $T_{\text{avg}}$), among all runs (except the cold runs) is around $\pm 15.6\%$ of the overall average temperature, regardless of the type of the temperature profile. This gives a $\pm 90$K variation from the overall average temperature of 528K among all runs in the Experimental Matrix. Similarly, for the radial profiles this variation is $\pm 16$K, and it is $\pm 45$K and $\pm 62$K for the uniform profiles and hot streaks, respectively. On the other hand, the circumferential variation for both radial and uniform profiles is $\pm 4$K, which was observed to be less than 5% of $T_{\text{avg}}$. This variation is higher for the hot streaks due to one of the upstream rakes located behind the struts for the un-cooled run. If this run is excluded, the circumferential variation comes down to $\pm 7$K in average, a comparable range to the other profiles. That is, the circumferential variation is clearly lower than the variation in the inlet bulk temperature.

A detailed statistical analysis of the data was performed and will be reported in depth in a formally documented manner at a later date. Within that effort, a statistical model was used to establish the 95% confidence limits on the temperature profiles. Statistical modeling allows data from non-repetitive conditions as well as the repeat conditions to be used by empirically modeling the variation in major parameters of interest such as mass flow or total inlet temperature. The larger set of data used in modeling results in more accurate statistics for even sparse data. How good the model reproduces the data is defined by 95% confidence limits.

The comparisons performed between the data and these predictions are reproduced here. In all figures, the average points show the average of the repeat runs over the two rakes. The range bars show the maximum-to-minimum range from the average for the experimental data, and 95% confidence limit for the statistical data. Since the radial profile is of more interest, this comparison focused on the radial inlet
temperature profiles. The data is given in the form of normalized values using the bulk average temperature of each run. The normalization accounts for the differences in the bulk average temperature, as well as the differences occurring due to different time windows, and gets rid of most of the variation among the runs. This comparison was performed at the time with different time windows that differed by ±10 ms in an average than the current ones used throughout the data analysis, however in a later section it will be shown that even a change of 28 ms in the time window still does not affect the heat transfer levels significantly, since the experimental operating conditions stay very similar. Hence, this comparison between the data and the statistical model can still be used in conjunction with the inlet temperature profiles demonstrated so far.

![Image of Figure 5.4](image-link)

**Figure 5.4 Comparison of Experimental and Statistical Averages for Radial Profiles**

Figure 5.4 shows all the radial temperature profile shapes at different cooling conditions. The same level of cooling is shown with the same color. The statistical model results look very similar to the experimental data. The experimental results use only the repeat runs in each profile shape, but the statistical model uses all available runs for
calculations, resulting in dramatically improved confidence limits distributed over different span locations. The main interest is in the un-cooled, nominally cooled, and high cooling cases.

Figure 5.5 Comparison of Experimental and Statistical Averages for Radial Profiles for the Main Conditions

Figure 5.5 shows these three main conditions specifically. This time, a direct comparison between the experiment and the model is presented with solid and empty circles, respectively. It is clear that the average inlet temperature profiles are very similar. The model always captures the experimental data average within the confidence limits.
Other than the un-cooled case, the main difference is in the range bars. The 95% confidence limit for the statistical model is much smaller than the range bars in the experimental data, since it uses many more runs than the experimental data, which uses a few runs only. This is the primary benefit of statistical analysis, as the confidence limits for even a single run can be predicted using all available runs. The CFD code developers can use these confidence limits to help quantify the effects of inlet variation on the prediction produced by the code.

Figure 5.6 shows a comparison of the calculated 95% confidence limits and the range bars on the experimental data, and the 95% confidence limits on the statistical model. The total variation shown on the y-axis is the total length of the range bars used for each condition at each span. The same colors show the range bar definition, and the patterns show the cooling conditions. The bars with blue colors are always much larger than those with red colors, that is, the 95% confidence limits based on a standard deviation of the experimental data (blue) suggest much larger variations than the actual measurement repeatability (red). The 95% confidence limits on the experimental data

85
presented here are calculated approximately in order to help visualize the differences between the confidence limits of the experiment and the model on the same scale. In reality, they are much larger. The range bars based on the repeatability variation (red) are closer to 95% confidence limits defined by the statistical model (green). Specifically for the un-cooled condition, range bars are shorter than those of the model. This shows that in the absence of a full statistical model of all the data, the range of data typically used as a measure of repeatability in this report is closer to the standard statistical 95% confidence limits used when there are wide variety of samples, and that using a 95% confidence limit based on a standard deviation when there are fewer samples over estimates the uncertainty of the average value. A similar comparison was performed for the temperature measurements, and heat flux and Stanton Numbers on the airfoil surfaces and platforms of the rotor section as a part of this effort. The statistical model predictions agreed well with the experimental data. However, the same type of a comparison has not been performed yet for the vane data.

5.1.3 Measured Inlet Pressure Profiles for Experiments

Barringer et al. [54] analyzed the effects of different inlet pressure and temperature profiles on vane heat transfer. It was observed that the secondary flows become stronger depending on the inlet pressure profile shape, and affect larger areas within the vane passage. However, the inlet profile generator used for mimicking an aero-engine combustor in [54] utilizes an injection technique to provide several streams of different temperature flow. This injection process disturbs the pressure field, resulting in different inlet pressure profiles. In the current research, a passive heat exchanger technique is used rather than injection, smoothing out the flow, and creating a uniform inlet pressure field.

Figure 5.7 shows the pressure distribution upstream of the vane section. There is one total pressure rake located at the same plane with the two total temperature rakes. Each data point shows the average normalized pressure of the runs with the same temperature profile regardless of the cooling condition. The range bars show the peak-to-peak variation (maximum-minimum) among all runs within that profile.
The difference between the cold runs (4 runs) and all other runs with radial, uniform and hot streak profiles (32 runs), are more observable. However, the variation within all these 32 runs is only 0.31% of the normalized pressure, $P/P_{avg}$, and the overall variation across the whole experimental matrix is only 0.6%. This is the average percentage variation from the average normalized pressure at all spans, and shows that the inlet pressure profile of each run is very closely replicated, and the variations are indeed very small.

### 5.2 Data Quality

As mentioned in Section 4.3, the vane is heavily instrumented with double-sided Kapton heat-flux gauges at the inner endwall, at 5%, 15% 50%, and 90% spans on both the suction and the pressure surfaces of the airfoil to obtain a detailed heat transfer map over the whole vane section. As previously stated, a primary goal of this research is to observe how the inlet temperature profiles and cooling levels affect vane heat transfer. Because the changes in input conditions are often small (e.g. change in mass flow rates between conditions of only 2% per Table 5.2), quantification of the measurement accuracy is critical to interpret the data correctly.
Sources of uncertainties vary between measured and calculated parameters. For measured parameters these come both from repeatability of the test conditions, but also uncertainties arising from the calibration and the processing techniques used during analysis (generally called instrument precision errors). The relative importance of repeatability versus the instrument precision will be discussed, but in general the instrument precision errors were found to be much lower than the measurement repeatability for all sensors. Thus, the repeatability, being larger, is used throughout the data analysis as an indicator of the overall accuracy of the measurements. These are then used via traditional error propagation techniques to create the uncertainty in the calculated parameters such as Stanton Number.

5.2.1 Fundamental Measures and Repeatability

The fundamental measurements: temperature, pressure, and heat flux have had the instrument precision quantified many times. The major design goal for these types of experiments is to drive the individual precision of any given instrument, below the repeatability of the test conditions, so that instrument errors do not dominate. We can take further advantage of multiple measurements to yield better estimates of the mean values.

Table 5.4 gives the variation in inlet conditions to the turbine for each repeat group of runs within the experimental matrix. The repeat groups are mostly performed for radial runs at all cooling levels, but there is also one set of repeat runs for the uniform profile. Of these repeat groups, Runs 18 and 19 do not have purge flow, while Runs 40 and 41 do not have vane outer cooling. These groups will be of interest in the upcoming sections. Group of Runs 43 and 47 is the case where there was no coolant supplied at all. The variations in all inlet conditions are generally less than 1% of the average value. This shows that the repeatability of the average for these conditions is very good. The following sections discuss in a little more detail the relative importance of instrument precision and repeatability by major instrument type.
<table>
<thead>
<tr>
<th>REPEAT RUNS</th>
<th>Temperature Profile &amp; Cooling Level</th>
<th>$P_{0,inlet}/P_{0,outlet}$</th>
<th>$P_{0,inlet}/P_{avg}$</th>
<th>$T_{0,inlet}/T_{avg}$</th>
<th>$\dot{m}$, Inlet Mass Flow Rate, kg/sec</th>
<th>$N_{corr}$ (rpm/$\sqrt{K}$)</th>
<th>%Vane Inner Coolant/Total</th>
<th>%Vane Outer/Total</th>
<th>%Purge/Total</th>
</tr>
</thead>
<tbody>
<tr>
<td>17,23,24</td>
<td>radial, low</td>
<td>4.51±1.59%</td>
<td>1.00±0.00%</td>
<td>0.98±0.00%</td>
<td>14.599±2.00%</td>
<td>367.72±1.66%</td>
<td>4.58%±0.11</td>
<td>4.29%±0.10</td>
<td>1.64%±0.04</td>
</tr>
<tr>
<td>18,19</td>
<td>radial, low, no purge</td>
<td>4.52±0.08%</td>
<td>0.99±0.00%</td>
<td>1.00±0.00%</td>
<td>14.334±0.02%</td>
<td>356.44±0.09%</td>
<td>4.72%±0.03</td>
<td>4.43%±0.03</td>
<td>0.00</td>
</tr>
<tr>
<td>27,28</td>
<td>radial, nominal</td>
<td>4.61±0.59%</td>
<td>1.01±0.00%</td>
<td>0.98±0.00%</td>
<td>14.155±0.07%</td>
<td>371.49±0.06%</td>
<td>6.93%±0.03</td>
<td>5.40%±0.02</td>
<td>0.70%±0.00</td>
</tr>
<tr>
<td>29,30</td>
<td>radial, high</td>
<td>4.56±0.05%</td>
<td>1.01±0.00%</td>
<td>0.99±0.00%</td>
<td>14.018±0.76%</td>
<td>369.39±0.45%</td>
<td>8.06%±0.00</td>
<td>6.28%±0.00</td>
<td>0.82%±0.00</td>
</tr>
<tr>
<td>40,41</td>
<td>radial, nominal, no vane outer</td>
<td>4.56±0.27%</td>
<td>1.00±0.00%</td>
<td>0.96±0.00%</td>
<td>14.574±0.37%</td>
<td>373.62±0.64%</td>
<td>7.08%±0.04</td>
<td>0.00</td>
<td>0.72%±0.00</td>
</tr>
<tr>
<td>43,47</td>
<td>uniform, un-cooled</td>
<td>4.68±0.41%</td>
<td>1.00±0.00%</td>
<td>1.12±0.00%</td>
<td>14.472±0.08%</td>
<td>377.80±0.19%</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>44,45</td>
<td>radial, nominal</td>
<td>4.58±0.44%</td>
<td>1.00±0.00%</td>
<td>0.95±0.00%</td>
<td>14.412±0.02%</td>
<td>377.01±0.00%</td>
<td>6.76%±0.02</td>
<td>5.27%±0.02</td>
<td>0.68%±0.00</td>
</tr>
</tbody>
</table>

Table 5.4 Repeatability of Experimental Input Conditions For Different Groups
5.2.1.1 Inlet Total Temperatures

Both the inlet and exit total temperatures were calibrated extensively using a total temperature blowdown rig after the experimental matrix. This accounted for individual sensor variation, but during the matrix the calibration values did not change for individual sensors. This implies that any bias effects would be constant, and thus the relative changes between runs would not be affected. This produced very good repeatability of average temperatures. For example, for the radial nominal cooling case the variation between Runs 27 and 28 is only 0.02%, which corresponds to a ±0.1K variation in average inlet temperature, while a 0.27% variation for radial low cooling case (Runs 17, 23, and 24) means only ±1.43K. This suggests that the inlet temperatures in this series of experiments are very stable. The variations of the inlet bulk temperatures between repeat runs for low and nominal cooling levels for radial profile and the un-cooled case for uniform profile stay within ±1.5K. For high cooled radial profile this variation is ±4K, and for nominal cooling with no vane outer it is ±8K. For the cooling gas temperatures, the flow rates are low enough that no total temperature calibration was required, although they use the same general layout as the inlet total temperatures.

5.2.1.2 Pressure

The Kulite pressure transducers had calibration accuracy of ±0.2 psia that corresponds to a ±1.5 kPa. For different repeat groups, the variation in inlet total pressure differs. For example, for low cooling case (Runs 17, 23, and 24), this variation is ±9kPa, while it is as low as ±0.27 kPa for the nominal cooling case (Runs 27, 28). That is, the precision accuracy is worse than repeatability of the measurements. However, the pressure measurements are only referred to at the inlet. The analysis in this thesis focuses on heat transfer, and this accuracy issue does not affect the data presented.

5.2.1.3 RTD (Resistance Temperature Device)

Standard reference temperature sensors were installed throughout the rig to provide a calibration standard for HFG comparisons. The sensors used for this study were located in the vane cooling cavities on the inner bulkhead separating the inner and outer cooling cavities. These sensors were all calibrated before the experiments in an oil bath.
accurate to ±0.1 deg C. Typical calibrations accuracies through the A/D system are about ±0.3 deg C.

### 5.2.1.4 Temperature and Heat-Flux Measurements From Kapton Gauges

For the Kapton heat-flux gauges, the precision of the measurements depends on several factors. There are several substrate properties: mainly thermal conductivity, \( k \), and the thickness, \( d \), but there are other secondary ones such as the thin-film resistance, and the scale factor. The uncertainties in the substrate properties and the measured resistance can be shown to be quite small leaving the main error sources being the initial temperature and the uncertainty in the gauge scale factors.

![Figure 5.8 Variation in RTD Measurements](image)

The initial gas temperature is based on individual RTD temperature measurements at each vane, as there was some circumferential variation that could be accounted for. This created an uncertainty of ±1% on average (around ±3K). This variation is shown in Figure 5.8. Each data point is the average of all RTD measurements from the vane airfoils with the range bars showing the range (maximum-minimum). The different
regression techniques used resulted in a stability value on the order of ±1%. This means a variation of around ±2.5K in the calibration for a typical resistance value of 250 ohms. After the calibration was applied, the typical upper and lower gauge temperature variation was found to be around ±1K.

Given these inlet conditions, the variation of the measurements obtained from heat-flux gauges at the target point would be interesting to know. The average variation for all the upper and lower gauges (regardless of location) for radial nominal cooling repeat case (Runs 27, 28) is only 0.32% of the average temperature measurement among repeat runs, while the corresponding variation for radial low cooling case (Runs 17, 23, and 24) is 0.45%. The variation in these gauge measurements is slightly higher than the measurements from the rakes. This is a result of the random variation of each gauge adding up to the variation at the inlet conditions, which means that the variation of each gauge is even much tighter than these values above. Figure 5.9 shows this variation for upper and lower gauges for the radial nominal cooling case among repeat Runs 27 and 28. The variation in temperature measurements for each gauge stays below 1.4% of the average temperature of the repeat group at all span locations. This corresponds to a ±5K variation at most. However, the typical variation is around ±1K. Each data point shows the variation among these runs as half of the range (maximum temperature measurement minimum temperature measurement) divided by the average temperature measurement of repeat runs.
Figure 5.9 Repeatability for Nominal Cooling
The effect of these values on the deduced heat flux can be shown through the simplified steady-state model of heat flux that examines the uncertainty in heat flux as a function of k/d and the difference between the upper and lower temperatures. The simplified version of equation (26) yields:

\[ q'' = -\frac{k}{d} \Delta T \]  

(28)

The k/d remains constant, and the same gauge is compared between different runs. The direct propagated variation in heat flux varies directly with the uncertainty in temperature. However, since both the upper and lower gauges are set to the same initial temperature at the beginning of a run (via the RTD temperature), this variation reduces just to the effect of the scale factor between the two sensors, which is about \( 2^{1/2} \times 1\% \). Thus, propagated errors in heat flux should be on the order of ± 1.5%.

Although the variations in temperature measurements are very small among repeat runs, the heat flux calculations give significantly larger variations. Generally, there is a ±4000 W/m² variation among repeat runs on the airfoil surface. Figure 5.10 shows each repeat run with a different color. For a typical heat-flux measurement of 50000 W/m², this corresponds to 8% variation. At the inner endwall, these values are ±5000 W/m² variation out of 7000 W/m², and yield a 70% variation. These are the measured experimental repeatability, and are much larger than the propagated error quoted as ± 1.5%. This implies that there are other conditions that are dominating the uncertainty. Thus the repeatability will be quoted as the dominant uncertainty, rather than the precision error.

As an additional note, in Data Processing, it was mentioned that the filtering process causes a reduction by 0.02% in the time-averaged temperature values, and an increase by 0.05% in the raw heat-flux measurements. This variation is very small compared to the variations in heat-flux measurements and Stanton Numbers.
Figure 5.10 Typical Heat Flux Values and Repeatability
5.2.2 Calculated Stanton Number

The Stanton Number is a normalized heat flux, where the effects of driving temperature and mass flow are accounted for with the goal of discarding these differences in the vane inlet conditions for each run, and as defined in equation (9):

\[
St = \frac{h}{\rho \cdot \alpha \cdot C_p} = \frac{q''}{m \cdot \frac{A_{ref}}{A_{ref}}} \left[ (C_{pT})_{ref} - (C_{pT})_{w} \right]
\]  

(9)

Equation (9) shows that Stanton Number normalizes the measurements with the driving temperature and the mass-to-area ratios, therefore, reducing the variations in the flow parameters introduced at the turbine inlet such as the inlet bulk temperature, the cooling amounts, and the reference areas. This provides a more accurate comparison between different cases. Hence, comparing heat flux distributions directly does not help to isolate the influence of temperature profile alone, and drawing conclusions could be misleading. Normalizing heat flux by these varying parameters (using Stanton Numbers) is the key to understand the root cause of the differences. If the variations in heat flux are due to these effects, the data points in terms of Stanton Number should collapse significantly. Further discussion regarding the derivation of Stanton Number can be found in the Appendix.

The Stanton Number definition used in the data analysis utilized the “local” upstream temperatures from the closest rake to the instrumentation location for the majority of the runs to normalize surface heat flux values, rather than utilizing the average upstream temperature, to reflect the measurements more accurately. 50% and 90% span data used the corresponding local measurements from the closest upstream rake. For 15% span, the average, measurement from 10% and 20% of the closest rake was used, since there is no thermocouple located at 15% span on the rakes. Similarly, to normalize the data obtained at the endwall and 5% span, the 10% span rake measurement was utilized, as this is the last available measurement location on the rake that is closest to the hub. For hot streak runs, the alignment of the hot streaks with the downstream vane section was varied. This caused the upstream rakes to be exposed to different sections of
the hot streaks during the runs. However, the 37.5% pitch of the heater sector was always aligned with one of the rakes. Therefore, instead of the local measurements from the closest rake to the airfoil instrumentation, the local measurements from the rake that is aligned with the 37.5% pitch of the heater sector was used for normalization to avoid the influence of the alignment variation in the comparisons. Many different comparisons were done using Stanton Number definitions derived from various temperature locations on the rakes and using different extrapolations to the endwall. No significant changes in the Stanton Number distributions were observed, although the levels did shift somewhat with choice of reference temperature. Thus the main results shown are not specific to the choice of the reference temperature location, as long as it consistently represents the profile effects across different runs.

The repeatability in Stanton Number for a set condition is similar to that of heat flux. There is a general variation of $\pm3\times10^{-5}$ among repeat runs on the airfoil surface, which means about 10% of average variation for a typical Stanton Number of 0.0003 (Figure 5.11). A comparison of the variations between the heat-flux and Stanton Numbers shows a more normal distribution on the Stanton Number, indicating that some of the variations in heat-flux have been accounted for, but in general they are quite similar. This variation is about $\pm8\times10^{-5}$ at the inner endwall, which is of 80% of average variation for a typical Stanton Number of 0.0001. The percent variations at the endwall are much higher because of relatively lower heat transfer levels. These variations are however still much smaller than the variations between different conditions of cooling levels and temperature profiles, which enables to resolve all these differences as will be shown in data presentation.

Since the percent variations in Stanton Numbers among repeat runs are much higher than the uncertainties caused by error propagation through the measurements among these runs, the range bars in the presented data will represent the variation in Stanton Numbers. The variation is presented in terms of absolute difference from the average rather than the percent variation.
Equation (9) shows that in addition to the heat flux and the mass-to-area ratio, the reference and wall temperatures and their corresponding specific heat values also play a role in the derivation. \((T_{\text{ref}} - T_w)\) was previously defined as the driving temperature. Here, \(T_{\text{ref}}\) is defined as the inlet rake temperature, and \(T_w\) as the Kapton upper gauge
temperature. From this point on, it will be replaced with $T_u$ to be more specific. Considering that the instrumentation is spread on different spans of eight airfoils, the closest rake to this instrumentation and the local rake temperature corresponding to the particular span of interest is used. The hot streaks were not aligned exactly with either of the two rakes. However, one of them was always aligned with the 37.5% pitch of the heater sector. Therefore, that rake is used for the hot streak runs rather than the closest rake. All the parameters shown in equation (9) add up to the overall propagated uncertainty in Stanton Number. The uncertainties for the heat flux calculation and the mass-to-area ratio result in a direct linear addition to the overall uncertainty in Stanton Number. The overall uncertainty in Stanton Number is defined with equation (29):

$$\frac{\Delta St}{St} = \left[ \frac{\Delta q}{q} + \left( \frac{\Delta m}{m} \right)^2 + \left( \frac{\Delta m}{m} \right) \left( \frac{\Delta T}{T} \right)^2 + \left( \frac{\Delta T}{T} \right)^2 + \left( \frac{\Delta T}{T} \right)^2 \right]^{1/2}$$

The main focus here is on the influence of the driving temperature on the Stanton Number uncertainty. For this purpose, ignoring the uncertainties carried over by the heat flux and mass-to-area ratios, the typical uncertainty analysis would yield the following equation:

$$\frac{\Delta St}{St} = \left[ \left( \frac{\Delta C_p}{C_p} \right)^2 + \left( \frac{\Delta T}{T} \right)^2 \right]^{1/2}$$

Equation (30) shows the uncertainty in the Stanton Number as a function of the uncertainties in the temperature measurements and the specific heat values.
In the data analysis, \( C_p \) is evaluated as a function of both the inlet pressure and the inlet temperature that were described above. For the flow conditions of interest here, \( C_p \) can be defined to be a function of temperature only. This results in a variation of about \( \pm 0.02\% \) in \( C_p \) values. The rake measurements showed that the circumferential uniformity is satisfied within \( \pm 5\% \) of \( T_{avg} \). If only the repeat run inlet conditions are considered, this variation is less than \( \pm 1.5\% \) of \( T_{avg} \). The last measurement location available at the rakes is 10\% span. This measurement was used in Stanton Numbers presented for 5\% span and endwall data. If, however, there were thermocouples enabling the corresponding 5\% span and endwall measurements at the rakes, the local rake temperature measurement variation would go up to \( \pm 2.5\% \). This observation comes from predicted temperatures at these locations via extrapolation. This procedure and the influence on the Stanton Number of these local measurements instead of 10\% rake measurements will be a topic to be discussed in Appendix. This amount of a temperature change will cause an additional variation of \( \pm 0.14\% \) in the corresponding \( C_p \) value. Including 11 repeat runs, the average variation in the upper temperature measurements from the same gauge that were in pairs and used for Stanton Number calculation is \( \pm 0.58\% \). Using these variations, the uncertainty in the Stanton Number calculations with respect to the driving temperature can be analyzed graphically based on many runs, as shown in Figure 5.12.

The driving temperature (not unexpectedly) is the main driver in the Stanton Number uncertainty. The numbers show different runs having different profile shapes and different cooling levels. However, the driving temperature (the difference between the upstream rake temperature and the local fluid temperature) tends to fall within a range for each run. The addition of heat flux and mass-to-area uncertainties to the overall uncertainty are just linear additions. As long as this difference stays higher than 50, the uncertainty level is fairly low - lower than 20\%. If this difference can be doubled, the uncertainty will be halved. However, if this difference cannot be maintained, the uncertainty starts behaving exponentially and shoots up once past below 50.
This finding suggests that to ensure reliable Stanton normalization, the temperature difference introduced by the inlet temperature profile should be adjusted in such a way that the temperatures at the inlet stay higher enough than the corresponding local fluid temperatures on the airfoil surface. For example, Figure 5.12 indicates that the Stanton Number uncertainty is the highest for Runs 20, 35, 42, and 46, all of which are before the point 50. Run 20 is a cold run, and the same trend is observed for the other cold runs that have a very low temperature difference. Run 35 is a hot streak run aligned with mid-passage. Run 42 and Run 46 are also hot streak runs. Run 46 is the only uncooled hot streak run that will be used in the remaining part of the analysis section. The driving temperature on Run 31 for instance is quite large, creating a relatively accurate
calculation of the Stanton Number, while Run 46 has essentially no driving temperature
difference (in fact it is close to an adiabatic wall temperature run) and thus the heat-flux
and Stanton Numbers are quite close to 0, and the relative uncertainty is quite high. The
small temperature differences correspond to the inner endwall region, which also includes
5% span. It is expected to have this situation arising, since the temperature profiles have
large gradients at the inner wall, especially for the hot streak runs. Comparison between
different conditions is usually done through a Stanton Number Reduction Factor
calculation, which just is a linear combination of two different Stanton Number
calculations, so there is not detailed analysis needed.

5.2.3 Summary

In this chapter, the experimental matrix and the experimental operating conditions
of the high-pressure turbine stage have been introduced, and the data quality was
evaluated. There are many variables considered in this discussion such as inlet pressure,
mass flows of the core and coolant flows, inlet temperature profiles, and corrected speed
as listed in Table 5.2. The controlled variables among this list are the inlet temperature
profiles and the coolant mass flows. Hence, the analysis of these two variables will be
performed in great detail in Chapter 6. All these inlet variables serve as initial checks in
establishing confidence in the measurements throughout the vane section.

The temperature profiles showed that the circumferential uniformity is satisfied
generally within ±5% of $T_{avg}$, and is even tighter for the uniform profiles. This confirms
that the primary requirement of generating uniform and radial profiles, the
circumferential uniformity, has been established. The profile shapes reflect temperature
gradients that exist for radial profiles and hot streaks, while the uniform profiles reflect
their radial uniformity clearly. A statistical model used in conjunction provided the 95%
confidence limits on the data. It was shown that the variation in the experimental data,
when represented via range bars, was similar to the 95% confidence limits as predicted
by the model, rather than a use of the 95% confidence limits of the data. Hence, the range
bars will be used in representing the variation in the conditions instead of the standard
deviation, whenever available. For the inlet pressure profiles, the variations throughout
the experimental matrix are measured to be very small. Since the heater used a passive
heat exchanger technique, the pressure profiles created were uniform at all times, and did not interfere with the heat-flux measurements obtained on the vane surfaces.

There are several measurements performed to define both the inlet (or operating) conditions of the high-pressure turbine, as well as the heat transfer through the high-pressure vane. The quality of this data is critical for the accuracy of the results and proper interpretation in the analysis. Thermocouples, pressure transducers, and heat-flux gauges are all shown to have good repeatability in the measurements they provided. This tight repeatability is reflected onto the calculated Stanton Number presented with the heat flux in the next chapter.

This comprehensive view of various measurements performed during the experiments sets the operating environment of this particular engine under investigation. All the variation sources are identified, and the variations in the measurements that will be shown with range bars throughout the analysis in Chapter 6 are detected to be due to these sources occurring at the inlet, and not due to the uncertainty in the measurements arising due to the measurement devices, since the repeatability in these measurements was found to be much smaller than the variations between different inlet conditions.
CHAPTER 6
DATA ANALYSIS

In an actual engine, the first stage vane is exposed to high turbulence levels and temperatures due to its location downstream of the combustor exit. The interactions of cooling gas injection, inlet temperature profiles, endwall effects, and vane/blade interaction make the flow physics associated with the vane row extremely complicated.

A primary experimental goal of this effort is to measure the effect of cooling levels, vane inlet temperature profile shape and their interactions on the high-pressure vane. Using the experimental procedure and basic data reduction of Chapter 4, the measurements are presented in both absolute and normalized terms. This provides both the engine designer and the computational code developers the data they require. The primary measurements obtained are temperature, pressure, and heat flux data for the many different sections of the high-pressure turbine vane row. Section 6.1 presents the basic data used in the temperature profiles and cooling level analysis presented later in this chapter. By addressing bulk average temperature effects and the effect of the gas ingestion during the un-cooled experiments, the more detailed analysis presented in the later sections benefits from increased spatial resolution where the effects of:

1. inlet temperature profile, and
2. cooling level

can be isolated and compared.

A wide range of inlet conditions was utilized for this project. Three different vane inlet temperature profiles were investigated: uniform, radial, and hot streaks. Understanding the effect of the temperature profile on vane heat transfer is important, as the temperature profile is specific to the combustor design used in an engine. Accounting for the driving temperature effects helps provide the designer guidance needed to obtain measurements acquired in this work and apply them to other turbine designs.
In order to investigate the influence of cooling level, different coolant amounts (none, low, nominal, and high) were supplied through different cooling circuits, which made it possible to observe the influence of the purge flow and the vane outer flow (mainly the trailing edge cooling) on the vane heat transfer.

The presentation of the data and the influence of the control variables is a detailed process, as comparing both the heat flux and the Stanton Number provide insights that either measure by itself would not. As the data is presented for both the designer and the code developer, different parts of the data are included for each group. Each section has detailed discussions about the effects present in that part of the analysis, but for both the designer and code developer there are some key findings that summarize the main efforts of this work.

**Key Findings:**

1) From the design perspective, the effect of cooling can be seen throughout the vane, but the reduction on heat transfer appears greatest on the hub and diminishes as one moves out the vane span towards the tip. The differences between cooling levels is most easily seen in the hub and 5% span region, especially on the pressure surface, and through the inner endwall passage. The span-wise variation that occurs is larger for cooled conditions than for un-cooled ones.

2) The effect of the vane outer cooling circuit on the vane trailing edge is measurable and points to an area, where even in an isothermal facility such as this, a more complicated conjugate heat-transfer analysis may be required. This is seen even on the suction surface where there are no cooling holes connected to the vane outer circuit. The effect is comparable to the effect of the highest cooling on the airfoil surface. A similar effect is seen at the inner endwall exit.

3) The purge cooling flow levels have no impact on the vane heat-transfer levels (airfoil or endwall surfaces) for the purge cooling flow magnitude used in this study (1% of core). As would have been anticipated, the vane cooling circuits are independent of the purge and the two systems can be designed to meet their own needs without concern about interactions at this level of purge flow.
4) The variations caused by cooling level, including the un-cooled case, are similar to the variations caused by vane inlet temperature profile shape on airfoil surface heat transfer. However, the profile effects cause higher differences in the heat transfer levels compared to the cooling effects throughout the endwall region. Thus, in the endwall region, the profiles and cooling levels interact with each other.

5) The heat transfer appears to be significantly lower on the suction surface (as compared to the pressure surface) when the un-cooled data from this data set (Build 2) is compared to the un-cooled data from a previous dataset (Build 1, similar geometry but no cooling holes). On the pressure surface the levels are comparable. As anticipated, there is ingestion on the pressure surface with ejection on the suction surface. For this specialized case, the un-cooled data on the suction surface is most likely not useful for a designer, but is valuable for computational comparisons. For the endwall, there is no sign of ingestion for the un-cooled cases.

6) Hot streak profiles (which vary in the circumferential direction, as well as the radial direction) when aligned with the vane leading edge lower the Stanton Number compared to hot streak profiles aligned at the mid-passage, for the mid-span suction surface. Depending on the alignment type, there is also a slight decrease observed through the inner endwall passage, while there is more of a significant increase at the endwall exit.

7) At the endwall, low cooling cases provide the largest range in heat transfer among all cooling levels, with some sections having the highest loads, and others the lowest loads. But the addition of cooling reduces this variation at the endwall, indicating that the extra cooling flows are either reducing the driving temperature in this region or changing the secondary flows.

8) The use of the Stanton Number, to account for variations in heat-flux due to gas properties and driving temperature differences that works well for different inlet temperature profiles in the un-cooled case, fails to make significant improvements in the distribution of the heat-flux data throughout the vane when cooling is applied. This suggests that allocating the effect of the cooling gas by bulk average properties does not work. This is not surprising as the amount of cooling influence that each part of the
airfoil experiences is determined by the percentage of cooling available at that point, rather than the total cooling mass flow, as accounted for in typical Stanton Number calculations.

6.1 Basic Data and Preliminary Comparisons

The focus of the discussion will now shift to documenting the averages and variation in the control variables, as that forms the basis for interpreting the data from a design perspective and forms the boundary conditions for those interested in code validation.

The vane heat transfer data analysis requires a description of the data as a function of the main control variables, i.e. inlet temperature profiles and cooling levels. This is done in Section 6.1.1, where both the inlet conditions and the data are quantified for the variations in control variables. To isolate the effects, this data is then parsed out, recombined and analyzed to address the specific issues involved with profile and cooling by location on the airfoil surface. At the end of the chapter, a comparison of the main effects is shown.

The general data trends for different inlet temperature profiles at the same cooling level (which are available for nominal cooling and no cooling), and different cooling levels at the same profile (radial) are shown. Although there are three available temperature profiles for comparison (uniform, radial, and hot streaks), the hot streaks may have different influence on vane heat transfer depending on the alignment with the vane airfoils, i.e. alignment with the vane leading edge (VLE), or with the mid-passage (MP). Therefore, they are grouped in two parts to isolate any possible alignment effects, and are introduced as separate entries in data presentation. First, the inlet temperature profiles, which set the boundary conditions to the high-pressure vane, are introduced. Then, the heat transfer resulting from these inlet conditions are provided at different locations of the vane.

The discussion starts with the presentation of the data used in the analysis of Section 6.2. A comparison of the un-cooled measurements with the nominally cooled version (Case A) will be given to show the direct impact of cooling on the heat transfer levels. It is important to remember that on the suction surface for the un-cooled case,
there is ejection that occurs from the cooling holes because of free stream gas being ingested from the pressure surface. Case A consists of individual runs having very similar inlet temperature profiles to that of the un-cooled case. The radial profile makes use of two repeat runs. It is observed that the general trends of the data through the vane section clearly replicate the profile shapes created at the inlet. There is a pronounced reduction in high heat transfer levels observed with the uniform profile with the addition of cooling, which seemingly decreases the differences between the two profiles under investigation compared to the un-cooled case. Later, Case B is introduced, which is a combination of runs having relatively large inlet bulk temperature differences within a given profile. This comparative presentation aims to show that the differences between the two cases are very small. Hence, in the analysis of the temperature profile effects, Case B will be used since the addition of the extra runs helps in the data interpretation. The inlet temperature profiles and the data used in the analysis of Section 6.3 will also be demonstrated in this section. Any amount of coolant shows its effect as an immediate decrease in heat transfer at all locations, but the effects of coolant amount variation are not clearly observed right away. The ingestion effect through the pressure surface cooling holes is discussed via a comparison of Build 1 and Build 2 data, both on the airfoil surface and at the inner endwall.

The variety of different cooling levels offer a large set to choose from when starting the analysis of the influence of vane inlet temperature profile. The cases should be chosen in a manner that the only variable is the inlet temperature profile shape, while all the other inlet conditions remain constant. In order to remove any possible influence of cooling variation, the un-cooled runs could be chosen for a comparison. The uniform profile originally had two un-cooled runs (43 and 47) as stated in Table 5.1, but unfortunately, amplitude problem on Run 43 gave few usable sensors, leaving only Run 47 for the uniform profile case. Besides, the un-cooled data obtained on the vane suction surface has reduced heat transfer levels compared to those that would be observed on a solid vane airfoil without an implemented cooling scheme due to ingestion occurring through the holes on the pressure surface. Therefore, drawing conclusions from a profile comparison solely based on the un-cooled data would be misleading. Keeping this in
mind, both the cooled and the un-cooled trends will be examined. As a matter of fact, there is only a small variation in nominal cooling level among all 17 runs as shown in Table 5.2, and this variation is removed via using Stanton Number.

<table>
<thead>
<tr>
<th>COOLING LEVEL</th>
<th>RADIAL</th>
<th>UNIFORM</th>
<th>HOT STREAKS - MP</th>
<th>HOT STREAKS - VLE</th>
</tr>
</thead>
<tbody>
<tr>
<td>UN-COOLED</td>
<td>22</td>
<td>43,47</td>
<td>46</td>
<td>----</td>
</tr>
<tr>
<td>$T_0,\text{inlet}/T_{\text{avg}}$</td>
<td>0.99</td>
<td>1.15±0.00%</td>
<td>0.83</td>
<td>----</td>
</tr>
<tr>
<td>COOLED (CASE A) NOMINAL</td>
<td>27,28</td>
<td>31</td>
<td>35</td>
<td>37</td>
</tr>
<tr>
<td>$T_0,\text{inlet}/T_{\text{avg}}$</td>
<td>1.00±0.0%</td>
<td>1.17</td>
<td>0.91</td>
<td>0.91</td>
</tr>
<tr>
<td>COOLED (CASE B) NOMINAL</td>
<td>27,28</td>
<td>31,32,33</td>
<td>34,35,36</td>
<td>37,38,39</td>
</tr>
<tr>
<td>$T_0,\text{inlet}/T_{\text{avg}}$</td>
<td>1.00±0.0%</td>
<td>1.07±0.02%</td>
<td>0.98±0.02%</td>
<td>0.97±0.01%</td>
</tr>
<tr>
<td>LOW</td>
<td>17,18,19,23,24,25</td>
<td>----</td>
<td>----</td>
<td>----</td>
</tr>
<tr>
<td>$T_0,\text{inlet}/T_{\text{avg}}$</td>
<td>1.01±0.00%</td>
<td>----</td>
<td>----</td>
<td>----</td>
</tr>
<tr>
<td>HIGH</td>
<td>29,30</td>
<td>----</td>
<td>----</td>
<td>----</td>
</tr>
<tr>
<td>$T_0,\text{inlet}/T_{\text{avg}}$</td>
<td>1.01±0.00%</td>
<td>----</td>
<td>----</td>
<td>----</td>
</tr>
</tbody>
</table>

Table 6.1 Experimental Parameters for Runs Presented For Temperature Profile and Cooling Level Effects

Table 6.1 shows the runs presented for each temperature profile entry. Each data point is the average of the available runs mentioned for each profile. All values are scaled with the temperature average of all entries mentioned in the table. For the un-cooled case, the data points for the radial and the hot streak profiles come from one single run, while the uniform temperature profile data points are the averages of two available runs. Case A, similarly, has single runs for all profiles, but two for the radial. Case B uses the average of three uniform profile shapes. Nominal and high cooling levels use repeat runs, while low cooling level covers six runs with varying purge conditions, as the amount of purge coolant is very small (less than 1%). Purge flow is not expected to cause a significant difference on the vane heat transfer level, which will be demonstrated later.
As was shown in Table 5.2, low cooling corresponds to 10% of the overall cooling, while nominal and high cooling levels correspond to 13% and 15%, respectively.

The radial cooled runs 27 and 28 are used in almost every section of the analysis throughout the comparisons. This group of runs are nominally cooled radial runs with all coolant channels supplying cooling, and is used in conjunction with the nominally cooled uniform group of runs to represent the radial profile effects. For the cooling level effects, it represents a nominally cooled case, while all the other cooling levels all have radial temperature profile. In analyzing the vane outer coolant effects, this group of runs is used one more time as the runs with vane outer coolant supply on, and are compared to the runs with a radial profile and no vane outer cooling.

6.1.1 Raw Data for Profile Effects (Cooled and Un-Cooled)

To compare the cooled and the un-cooled heat transfer results as a function of profile on the airfoil surface, the actual inlet profile shapes should be studied first to detect the main causes for the differences between the two cases. In Section 6.2, the temperature measurements performed at the upstream rakes were presented through normalization with $T_{avg}$ of both rakes, which is the inlet bulk temperature. Figure 6.1 shows the temperature profiles for the two cases, un-cooled and nominal cooling, from two different perspectives: the scaled measurements, and the normalized measurements. Since Stanton Number is defined as the normalization of the data with the local inlet temperature measurement, the surface measurements should directly reflect the actual rake measurements at the inlet. The range bars show the maximum-to-minimum range between the runs used for each profile. With the same reasoning the Stanton Numbers are defined, the temperature profiles using the measurements only from the closest rake to the instrumentation are plotted for uniform and radial profiles, and the rake that is aligned with the 37.5% pitch of the heater sector is used for the hot streaks to avoid the variations caused by the hot streak locations with respect to the upstream rakes. The first row shows a comparison of the un-cooled case with Case A in terms of scaled temperatures with the maximum temperature in the group of profiles. The second row gives the same profiles in normalized version with the average bulk temperature of the closest rake to the instrumentation on the airfoil surface.
In Figure 6.1, the first row indicates a reduction in the profile magnitude from uniform towards the hot streaks. The instrumented spans on the airfoils correspond to 5%, 15%, 50%, and 90% span locations at the rakes, indicating that the highest heat transfer levels at all instrumented airfoil surfaces should be observed with the uniform temperature profile. There are high temperature gradients observed at inner and outer spans for the radial and hot streak profiles. Case A is nominally cooled, and has very similar inlet temperature profiles to the un-cooled case, which allows for a comparison of the effects of profile shapes on the airfoil surfaces. The use of the scaled temperatures (scaled by the maximum temperature of all the groups) preserves the fundamental shape.
that would be observed when using the absolute temperatures. The second row shows a change in trends when normalized with the average temperature. This is due to the fact that the variations in inlet bulk temperatures are removed via normalization, leaving just the general profile shape relative to the average temperature for each profile group.

For all airfoil surface presentations, the primary layout used consists of four sections, each of which shows a different span put in order from the tip towards the hub. The x-axis represent wetted distance (WD) at the respective span. The positive portion of the x-axis represents the suction surface, and the negative portion represents the pressure surface. The location 0% represents the leading edge of the airfoil, while 100% and –100% represent the same location, which is the trailing edge where both surfaces merge together. Often a direct comparison will be made between the heat-flux and the Stanton Numbers, as that comparison shows how much of the variation seen in the heat-flux levels can be explained by mass flow and driving temperature effects.

Figure 6.2 and Figure 6.3 show the heat flux and Stanton Number distributions on the airfoil surfaces, first for the un-cooled, then for the cooled runs of Case A. The data points show the averages of all available runs for a given condition as stated in Table 6.1, and the range bars show the variation among these runs within the same profile. All data points have range bars on them, other than the conditions existing of a single run. However, for some data points with more than one run, the bars are actually so small that they are embedded in the symbols and are not visible. The measurements were performed at the locations on the airfoil surface as shown in Figure 4.11. This layout gives the set of gauges that could provide measurements starting from Run 15 and on.

In Figure 6.2, the pressure surface reflects an actual un-cooled environment, but since the existence of cooling holes on the vane leads to a situation where hot gas is ingested on the pressure surface and ejected on the suction surface as shown later, the data on the suction surface does not reflect the actual conditions that would be observed on a solid un-cooled vane airfoil. However, despite the reduced heat transfer levels on this surface, there is still a very clear difference between the uniform and the other profiles when un-cooled. Although the scales used for the heat fluxes and Stanton Numbers are different by orders of magnitude, the distributions look very similar.
In Figure 6.2 and Figure 6.3, the distributions show that the trends in Figure 6.1 are generally preserved for both cases at all surfaces: the heat transfer decreases from uniform profile towards hot streaks. Figure 6.4 and Figure 6.5 show the comparison of the un-cooled and cooled – Case A. Apparently the cooling influence has clearly reduced the strong dominance of the uniform profile. This is observable from the relatively reduced scales of heat flux values, and much closer trends between all profiles. Physically, both 90% and 5% spans are the closest locations to the inner and outer endwalls, which receive a large amount of cooling from the vane inner and vane outer cavities. The effect of cooling is clearly observed at 90% span. The higher magnitude of uniform profile does not exist anymore with the addition of cooling. Actually, there is almost no influence of the temperature profiles observed here. The generation of horseshoe vortices also contributes in heat transfer reduction by mixing the flow further and bringing it to a more homogeneous state.

Due to lack of measurements at the vicinity of the leading edge with the showerhead-cooling pattern, the trends in this region for different spans are not visible. However, there is an observable immediate drop in the continuous increasing trend of the Stanton Number observed on the mid-span suction surface in the throat region that is around 40% wetted distance. This indicates the presence of a local disturbance occurring in the vicinity of 5% wetted distance downstream of the last row of cooling holes located on this surface. During this period, the heat transfer level towards the trailing edge on the suction surface reaches three times the levels observed on the pressure surface for the hot streak profiles, while this value doubles for the radial profile. The only measurement available at 50% closest to the trailing edge suggests a possible continuous decrease further downstream. The pressure surface heat transfer gradually increases towards the trailing edge due to the acceleration of the flow.
Figure 6.2 Heat Flux and Stanton Number Distributions on the Vane Airfoil Surface With No Cooling
Figure 6.3 Heat Flux and Stanton Number Distributions on the Vane Airfoil Surface With Nominal Cooling – Case A
Figure 6.4 Heat Flux Distributions on the Vane Airfoil Surface With No Cooling and Nominal Cooling (Case A)
Figure 6.5 Stanton Number Distributions on the Vane Airfoil Surface With No Cooling and Nominal Cooling (Case A)
The data points for 5% span in the case of the hot streak inlet temperature profile are not presented in terms of Stanton Numbers due to increased uncertainty in the calculated Stanton Number when the heat flux values are relatively low. Stanton Numbers use the local temperature measurements from the rakes. However, this is the region where very low driving temperatures are observed due to the low inlet temperatures closer to the wall indicated by the gradients shown in Figure 6.1. Figure 5.12 explained this situation as growing uncertainties in Stanton Numbers as the driving temperatures get closer to 0 and the heat flux follows suit. For example, two of the runs shown in this figure are Runs 35 and 46, a hot streak run aligned with mid-passage and the hot streak un-cooled run, respectively. Another hot streak run that has very low driving temperature is Run 37, an alignment case with the vane leading edge, which is not shown in Figure 5.12.

For the endwall, in all figures used to present the heat transfer distributions, the top figure shows the axial variation of heat transfer level starting from the leading edge towards the end of the passage at the trailing edge. The 0% location refers to the leading edge of the vane section, and 100% refers to the trailing edge. The middle figure gives the circumferential variation across the pitch between the two neighboring airfoils. This is the section corresponding to the region at the front of the vane leading edge that extends to 23% of axial chord further front at the inlet. 0% and 100% show the leading edges of the two neighboring airfoils. The endwall exit at the bottom is also demonstrated with respect to the same leading edge coordinates to be on the same scale through the passage. Due to the turning of the passage, the layout of the instrumentation is such that all gauges at the aft of the trailing edge are mostly lined up at negative pitch-wise direction. This section continues at the aft of the passage for another 7%, and will be much more clear with the introduction of contour plots of the entire vane passage that will be presented later. As there is a two-dimensional distribution of the cooling holes at the endwall (both axially and circumferentially) the cooling hole locations cannot be presented in the axial or circumferential distributions alone. Hence, the contour plots will be made use of to view the influence of the hole location.
Figure 6.6 and Figure 6.7 provide the heat flux and Stanton Numbers for the uncooled case and Case A at the endwall region, while Figure 6.8 and Figure 6.9 make comparisons between these two cases. The trends of the profile effects on the heat transfer look similar at the endwall section of the vane to that on the airfoil surface. Uniform inlet temperature profile clearly results in much higher heat transfer throughout the endwall region. This increase reaches the highest level at the endwall exit for the uncooled case (Figure 6.6). The uniform profile results in higher Stanton Numbers (by up to 0.001) than the radial profile. For the cooled case presented here, the difference between the two profiles as shown in Figure 6.9 has lowered to a 0.0004 Stanton Number on average showing the strong effect of cooling. Because of this significant decrease in the vane thermal environment, in many other parts of the airfoil, but mostly at the endwall, the Stanton Numbers turn out to be very small, fluctuating around zero. A zero Stanton Number means no heat transfer; that is, the adiabatic wall conditions are reached.
Figure 6.6 Heat Flux and Stanton Number Distributions at the Inner Endwall With No Cooling
Figure 6.7 Heat Flux and Stanton Number Distributions at the Inner Endwall With Nominal Cooling – Case A
Figure 6.8 Heat Flux Distributions at the Inner Endwall With No Cooling and Nominal Cooling (Case A)
Figure 6.9 Stanton Number Distributions at the Inner Endwall With No Cooling and Nominal Cooling (Case A)
6.1.2 Effect of Bulk Average Temperature on Profile

Next, Case B (this series considers all of the different inlet temperature profiles with different bulk average temperatures) will be introduced. Since the bulk average temperature changes for runs with different inlet temperature profiles, one question that arises is that if this variation could be ignored for the runs with similar profile shapes (e.g. the runs with uniform profiles, radial profiles, or hot streak profiles). Figure 6.10 shows that the differences between the inlet temperature profiles have decreased for Case B. The uniform profile still has temperatures of the highest magnitude at the rake locations corresponding to the instrumentation on the airfoil surface (5%, 15%, 50%, and 90% spans). One factor calling attention to the temperature profiles for Case B is that the range bars are much larger for the uniform and hot streak profiles. Because the runs used for these profiles are not repeat runs, the range bars show the variation in the inlet bulk temperatures and not the uncertainty. The same format of Figure 6.1 is followed here regarding the demonstration inlet temperature profiles. The second row is the normalized temperatures with the average temperature of the closest rake. The trends are different due to the removal of inlet bulk temperature variations.

Even if the variation within the same profile shape is large, the resulting variation in heat transfer is much lower due to the cooling effect. This can be observed in the upcoming series of graphs showing both heat flux and Stanton Number distributions for Case B, and a comparison between Case A and Case B, given in Figure 6.11, Figure 6.12, and Figure 6.13. In the same manner, the distributions at the inner endwall are given in Figure 6.14, Figure 6.15, and Figure 6.16. There is some visible reduction in the uniform profile from Case A to Case B in heat flux, but this difference is considerably reduced when comparing Stanton Numbers. Generally, the differences between the two cases are even much smaller when compared to the levels of the un-cooled case. This will be demonstrated in the summary section 6.1.5.
Figure 6.10 Inlet Temperature Profiles With Nominal Cooling For Case A and Case B
Figure 6.11 Heat Flux and Stanton Number Distributions on the Vane Airfoil Surface With Nominal Cooling – Case B
Figure 6.12 Heat Flux Distributions on the Vane Airfoil Surface With Nominal Cooling For Case A and Case B
Figure 6.13 Stanton Number Distributions on the Vane Airfoil Surface With Nominal Cooling For Case A and Case B
Figure 6.14 Heat Flux and Stanton Number Distributions at the Inner Endwall With Nominal Cooling – Case B
Figure 6.15 Heat Flux Distributions at the Inner Endwall With Nominal Cooling For Case A and Case B
Figure 6.16 Stanton Number Distributions at the Inner Endwall With Nominal Cooling For Case A and Case B
Figure 6.17: Heat Flux and Stanton Number Distributions at the Inner Endwall for Uniform Runs
Following the heat transfer distributions shown here with respect to percent axial chord and percent pitch, the data is also shown in two dimensions via contour plots in Figure 6.17. The program used in generating these contour plots is QuikGrid – Version5.4 [115]. QuikGrid uses C++ programming language for the source code of the grid generator and the contouring. Each contour plot scans the entire endwall section, covering the endwall inlet, passage, and the exit. The measurement locations are designated with crosses. All measurements distributed over four airfoil endwalls are presented in one single passage. The cooling scheme is also shown in the layout used with the Stanton Number contours. The airfoil section shown in these figures is not to scale. The remaining of the grid where there were no measurements is filled in with interpolation. The sparse gauge distribution makes the contribution of data interpolation more emphasized. As a result, these figures are only intended to give an idea about the general heat transfer trends at the endwall, and trying to read too much detail into the results could be misleading.

Figure 6.17 shows the endwall contour plots for the three uniform runs, Runs 31, 32, 33 separately, of which the uniform profile of Case B consists. Both the heat flux and the Stanton Number distributions in this region imply that indeed there are no significant differences among these runs.

In Section 6.2, the differences between the profile effects will be analyzed in more detail. The data presented in this section for the un-cooled and the cooled cases will be used for that analysis. For the cooled case, Case B is used instead of Case A, since both cases are very similar with Case B providing more data points.

6.1.3 Raw Data for Cooling Effects

In Section 6.3, there are four different cooling levels studied for the radial profile shape. Table 6.1 gives the runs used for each case. The inlet temperature profiles for these cases are given in Figure 6.18. The absolute temperature trends are given with the scaled temperature plot. The second plot shows that the slight differences between each case collapse even further by eliminating the differences in the inlet bulk temperatures via normalization. The range bars shown are the peak-to-peak variation from the average of all available runs at a given cooling level. The profile shapes are so similar that the
differences observed in the airfoil measurements will be primarily due to the differences between cooling levels. Converting the raw heat flux data to Stanton Numbers serves to remove not only the variations due to inlet bulk temperatures in the measurements, but also the variations in the amount of coolant supply to the channels within repeat runs. Both heat flux and Stanton Numbers are given on the airfoil surface and at the inner endwall for different cooling levels in Figure 6.19 and Figure 6.20. This data will be the source of the analysis that is presented in Section 6.3. All data points presented have range bars. The ones that do not show range bars either do not have a range bar, or the range bars are too small that they are not visible through the symbols.

Figure 6.18 Inlet Temperature Profiles For Cooling Levels

The major takeaway from these figures is that coolant supply does reduce heat transfer significantly on the airfoil surface with any type of cooling level compared to the un-cooled case. There is a 60% reduction in Stanton Number on average for all cooling levels, and corresponds to a decrease of around 0.0003 in Stanton Number. However, the differences between the cooling levels are not immediately clear.

The 5% and 15% spans receive the most-pronounced effect of cooling. At 5% span on the pressure surface is the location where the high cooling is really showing its effect compared to the other cooling levels. However, in general, the differences between
the cooling levels are not very clear at first sight. The 15% span location has about an 80% reduction in heat transfer generally on both surfaces, and this amount is halved for 90% span. The 90% span location has less distinguished coolant effect in the showerhead region. The main reason for this reduction despite the additional outer endwall coolant effect is the radial profile migration occurring at the tip. The increase in the heat transfer level at the tip has been observed consistently in all heat transfer trends so far. It is difficult to draw a more insightful conclusion for 90% with the limited data, but at 50% and 5% spans, the help of cooling in reduction of heat transfer diminishes towards the trailing edges on both surfaces. This is likely due to the elevated heat transfer on both surfaces due to the acceleration of the flow.

The differences between different cooling levels are not very clear at the inner endwall. Unlike the airfoil surface, the un-cooled run does not have consistently increasing heat transfer compared to the cooled runs. Throughout the passage, both the un-cooled and all different cooling cases show an alteration in heat transfer levels – the order of the heat transfer keeps changing among all cases. This is due to strong secondary flow effects observed in this region, which becomes more complicated with the addition of the film cooling. Higher heat transfer with no cooling at the endwall inlet seems to be partially washed out near the exit, suggesting an interaction with the flow field associated with the downstream rotor for these surfaces very close to the trailing edge, in addition to the secondary flow effect.
Figure 6.19 Heat Flux and Stanton Number Distributions on the Vane Airfoil Surface For Different Cooling Levels With Radial Profile
Figure 6.20 Heat Flux and Stanton Number Distributions at the Inner Endwall For Different Cooling Levels With Radial Profile
6.1.4 Comparison to Build 1

As was mentioned earlier in the thesis, Build 1 is the previous measurement program for which heat transfer and surface pressure measurements were performed using an un-cooled version of the same turbine stage. The same rotor and blade row was used in this research, but the vane row was replaced with vane hardware that includes a detailed cooling hole scheme on the airfoils and the endwalls. The portion of the Build 1 experiments that will be mentioned here was performed in shock tube mode instead of blowdown. However, the measurements were found to be insensitive to the operation mode of the facility [67] by comparing them to another portion of the experiments that were realized using the blowdown mode. Later, heat transfer computational efforts gave good agreements with the measurements [68-69]. Since the combustor emulator was made operational starting with the Build 2 experiments, data from Build 1 reflects the effects of a uniform temperature profile only. In this section, the comparison of the uniform un-cooled data from Build 1 and Build 2 will be presented. The measurements from Build 1 are available at 15%, 50%, and 90% spans, and at the inner and outer endwalls. This makes the comparisons at 15%, 50%, 90% spans and the inner endwall possible.

Figure 6.21 shows the inlet temperature profiles used for the two measurement programs. The data points in blue show the averages of six runs from two inlet rakes for Build 1 for which the operating design conditions were matched with those of Build 2 un-cooled runs. The data points in red show the averages of two runs from two inlet rakes for Build 2. All the data points are scaled by the same maximum of all temperature measurements. The average temperature used for the normalization is the average of both rakes. The variations among all these measurements are presented with range bars. The smaller variations obtained at the inlet profile with Build 2 shows how the overall quality of the repeatability has generally improved over Build 1 data. The average value at 50% span rake location is the most differing measurement between the two. However, the range bar shows that some of the runs of Build 1 matched the temperature values better with the current one at 50% span.
Figure 6.21 Inlet Temperature Profiles for Un-Cooled Uniform Runs of Build 1 and Build 2

Figure 6.22 shows the comparisons between the two builds at three different spans. The data points for Build 1 are the averages of six different runs, while those for Build 2 represent one single run, as the heat-flux gauge output for the other un-cooled run was mostly pinned due to high inlet temperatures. The Stanton Numbers shown here are much higher than the general levels observed for the cooled data presented so far. This is because the vane inlet area is used as the reference area instead of the throat area, and the reference temperature is defined as the average of the temperatures from both rakes instead of the average from the closest rake to the instrumentation. These parameters are used whenever there is a comparison with Build 1 data. The general trends suggest a consistent decrease on the suction surface at all spans, while the limited pressure surface measurements are aligned with the previous data set. The predictions of Tallman et al. [69] performed on Build 1 data set do not suggest this type of a sudden decrease in Stanton Numbers.
Figure 6.22 Heat Flux and Stanton Number Distributions on the Vane Airfoil Surface for Un-Cooled Uniform Runs of Build 1 and Build 2
The two builds used the same rotor, and the vane was replaced with another one of the same geometry, but with cooling holes. That is, Build 1 used solid airfoils in the vane, while Build 2 used airfoils with cooling holes drilled in them. For the Build 2 uncooled runs, the coolant was simply not supplied. However, the holes were still there, available for ingestion flow. The comparison shown suggests an ingestion of the mainstream gas through the holes on the pressure surface followed by an ejection through the holes on the suction surface of the airfoil. Therefore, ejection of this flow reduces heat transfer levels.

The two cases presented here have the parameters such as pressure ratios, inlet temperatures, and the corrected speeds of the turbine all matched. It is known that varying the time window that determines the target point does not influence the experimental operating conditions significantly. As a demonstration, another time window is used to obtain different target point values for the current uncooled uniform runs, which are designated with black circles in Figure 6.22. The new target point is chosen to be 28 milliseconds earlier but with the same duration corresponding to two rotor revolutions. The comparison shows that there is indeed not a considerable effect due to different time window selections in the heat transfer levels.

There were no pressure measurements performed on Build 2 vane, but in Build 1, the vane was instrumented with Kulite pressure transducers in addition to the heat-flux gauges, and the static-to-total pressure ratio was measured at each span. Later, CFD predictions for a data set with larger Reynolds Number were performed resulting in good agreement with this data [69]. Figure 6.23 shows the static pressure measurements normalized with the average inlet total pressure for the conditions for which the Stanton Numbers were shown above in Figure 6.22. The bars represent the ranges among the measurements for all six runs at two rakes.

The curves at all spans show a significant decrease in the static pressure once passed beyond the leading edge towards the trailing edge of the suction surface. Beyond the throat area at around 40% wetted distance, this pressure ratio levels out. This sharp decrease does not occur on the pressure surface. It stays rather constant, followed by decay only when closer to the trailing edge. Normally, while the airfoil is cooled, the
blowing ratio increases starting from the leading edge towards the trailing edge on the suction surface due to this decrease in the external static pressure acting on the holes. When there is no coolant supplied through the plenum, the pressure surface of the blade has the highest external pressure and this can cause ingestion of the outside flow into the holes. The mass flow into the blade has to exit through the suction surface, which is at a lower pressure. This is why the available data on the pressure surface and the leading edge area show a good match with each other, but the suction surface does not.

Figure 6.23 Pressure Ratio Distributions on the Vane Airfoil Surface for Un-Cooled Uniform Runs of Build 1 (reproduced from data in [114])
Figure 6.24 Heat Flux and Stanton number Distributions on Combined Spans for Un-Cooled Uniform Runs of Build 1 and Build 2
A denser distribution of the data points would reveal this ingestion effect more clearly. For this purpose, all spans are combined together for both builds in Figure 6.24. The general heat transfer trend is very clear with Build 1 data. With the addition of 5% span data from Build 2, the collapse of the pressure surface data between the two builds becomes more pronounced.

The comparison of this un-cooled data with the cooled data will help to quantify the effect of ejection on the suction surface compared to the cooling effect in this region. In Figure 6.25, the inlet temperature profile shapes are given for the runs studied, in accompany to the surface heat transfer distributions given in Figure 6.26. The data on the suction surface indicates that the reduction caused by the ejection of the flow through the suction surface holes already causes half-way reduction in heat transfer compared to the cooled case. That is, even if no coolant is supplied, having cooling holes on the surface helps with decreasing the local driving temperature.

Figure 6.25 Inlet Temperature Profiles for Uniform Runs of Build 1 and Build 2
Figure 6.26 Heat Flux and Stanton Number Distributions on Combined Spans for Uniform Runs of Build 1 and Build 2

Figure 6.27 shows the comparison of the un-cooled data from the two builds at the inner endwall region. In contrast to the suction surface of the airfoil, the distributions show that there is no noticeable ingestion effect taking place in this region. There is a consistent increase through the passage. This observation is in agreement with also Haldeman and Dunn [63], and Barringer et al. [55]. Haldeman and Dunn [63] had also reported that the heat transfer levels observed at the endwall were comparable to those on the airfoil surfaces. Yet, similar observations can be made with the data presented so far.
Figure 6.27 Heat Flux and Stanton Number Distributions at the Inner Endwall for Un-Cooled Uniform Runs of Build 1 and Build 2

Figure 6.28 shows the heat transfer distribution shown in Figure 6.27 in the 2-D endwall layout. The data for Build 2 are much sparse compared to Build 1, and the gauge locations used in two builds are not the same. However, these contour plots still give some idea about the general trend and the levels of heat transfer. Build 1 contours shown here nicely match with CFD predictions performed by Tallman et al. [69]. The contours reflect the slight decrease at the endwall for the un-cooled Build 2 data, but this is not anywhere near the significant reduction observed on the airfoil suction surface where ejection is known to be a problem for such a comparison.
Figure 6.28 Heat Flux and Stanton Number Distributions at the Inner Endwall for Un-Cooled Uniform Runs of Build 1 and Build 2 as Contour Plots (Build 1 – Left, Build 2 – Right)
In summary, the comparison between the un-cooled experiments from two different builds points out to an important finding: To simulate the environment of an un-cooled airfoil, supplying no coolant is not enough due to the significant blowing of the ingested flow through the suction surface holes. To achieve this requires either use of a solid airfoil with no holes drilled on it, or to use a very small amount of coolant just enough to avoid this ingestion effect.

6.1.5 Summary of Preliminary Findings

The data that will be the source of the analyses in the upcoming sections has been presented so far. The raw data for the cooled and un-cooled cases, the effect of bulk average temperature on similar profiles, and the ingestion/ejection effects through the cooling holes when there is no coolant supplied were the topics under discussion. There are a few main observations made at each section, e.g.:

1) With the addition of cooling, the differences between profiles are reduced significantly compared to the un-cooled case.
2) The differences in bulk temperatures between the runs with the same profile shape do not cause significant differences in heat flux, and they are further reduced when normalized (in terms of Stanton Number).
3) The addition of cooling is clearly observed compared to the un-cooled case, but the differences between different cooling levels are not as clear at first sight.
4) When there is a cooling scheme implemented for an airfoil, there is ingestion in the pressure surface cooling holes when there is no coolant supplied, which results in reduction in the suction surface heat flux, while the wall temperature increases, which needs to be avoided.

Some further examples to these effects on the airfoil surface will be provided in this section via summary plots of the data that has been given. All spans are combined so that all measurements available for the particular runs of interest can be viewed to clarify these effects.

Figure 6.29 shows the inlet temperature profiles, uniform and radial, for cooled and un-cooled runs. These are the profiles for the runs that will be used to present the differences in the profiles with and without cooling. The uniform cooled run is the
uniform run of Case B with Runs 31, 32 and 33. The normalization in the inlet temperature profiles is performed using the average temperature of the closest rake to the instrumentation for each group of runs shown. The range bars are much shorter when normalized.

![Inlet Temperature Profiles for Uniform and Radial Runs With Cooling and No Cooling](image)

**Figure 6.29 Inlet Temperature Profiles for Uniform and Radial Runs With Cooling and No Cooling**

The resulting heat transfer distributions on the airfoil surface are shown in Figure 6.30. There are significant differences between the un-cooled uniform and radial runs. Compared to the un-cooled case, the differences between the cooled uniform and radial runs look much smaller. This is especially observed in terms of Stanton Numbers, after removing the variations in the mass flow rate, the bulk temperature, and the reference area via normalization. It will be soon shown that the variations caused by the profiles in heat transfer are at comparable levels for cooled and un-cooled cases. Therefore, it is only the Stanton Numbers getting smaller for the cooled cases.
For the temperature profile effects, the runs that were presented under the entry of Case B will be used. The uniform profile Case B consists of the runs that are not repeat (Runs 31, 32, 33), and that have large inlet bulk temperature differences as shown in Figure 6.31. Run 31 represents the uniform profile of Case A. The normalized temperature shows a collapse of the profile shape as expected, since normalization with the average temperature removes the bulk temperature differences.
Figure 6.31 Inlet Temperature Profiles for Uniform Runs of Build 1 and Build 2 With Cooling and No Cooling

Figure 6.32 Heat Flux and Stanton Number Distributions on Combined Spans for Uniform Runs of Build 1 and Build 2 With Cooling and No Cooling
Similarly, almost a collapse of the cooled uniform data for Case B with Case A is observable in Figure 6.32, especially in terms of Stanton Numbers. Since Build 1 data is used in this comparison, Stanton Numbers use the inlet area and the bulk average temperatures in normalization, which results in higher Stanton Numbers. Figure 6.32 not only shows the insignificance of the bulk temperature difference on the heat transfer, but also shows the effect of the ingestion occurring through the holes on the pressure surface. This figure would actually be a replicate of Figure 6.26, if there were no Run 31 comparison.

Due to small Stanton Number levels arising in the flow field with the addition of cooling, appreciating the differences between the profiles is not easy at first sight, which requires a detailed analysis as will be demonstrated in the upcoming sections. The differences between the effects of the temperature profiles can be visually observed at the cooled environment, but this is not as clear for different cooling levels. It will be shown that the percent variations between the cooled and un-cooled cases caused by varying the inlet temperature profile or the cooling level are all at comparable levels on the airfoil surface. At the endwall region, there are larger differences observed by variation in the temperature profile shape. All these effects summarized here serve to establish the main characteristics of the data in hand, and will be utilized throughout the data analysis.

6.2 Temperature Profile Effects on Vane Heat Transfer

Inlet temperature profile has a significant influence on both the local aerodynamics and the heat transfer within the turbine. Being the closest section to the combustor exit, the first stage vane receives the largest effects from the inlet profile shape. With a large source of data as recorded in this current research with film cooling, the influence of different profiles can be examined at more than one span location on the vane surface, also making a direct comparison of span-wise heat transfer a possibility.

Among the temperature profiles examined in this study, uniform and radial profiles are both circumferentially uniform profiles, while hot streaks are not. There are more variables going into establishment of hot streaks, which distinguish them from the other two, as will be explained later. In the first two sections, the focus will be on the uniform and radial profiles, and on their similarities and differences. The first section is
dedicated to the effects of the profiles and the variations between them on the airfoil surface, while the second section presents the analysis at the inner endwall. Hot streaks will be the discussion topic of the last section on temperature profile effects.

6.2.1 Temperature Profile Effects on Airfoil Surface

In the following pages, the profile effects will be demonstrated in a different fashion than has been done so far. In order to distinguish the effect of span locations, each profile is split into four different spans, which are color-coded. This approach helps to isolate the main sources of the variations. As there are four instrumented spans, the span-wise variations can also be quantified. Both the absolute and percent deviations are given leading to comparative analysis between different cases. These are the comparisons of the heat flux and Stanton Numbers, the cooled and the un-cooled data, and the variations of profiles between each other and between the spans. Both cooled and un-cooled data presents the comparisons of heat flux and Stanton Numbers, as well as the comparisons between the profiles and the spans. Cooled data analysis is given in Figure 6.33 to Figure 6.35. Following this analysis, the un-cooled data analysis is given in Figure 6.36 to Figure 6.38. After establishing an understanding of the cooled and the un-cooled environment effects on the temperature profiles, the focus is switched to the similarities and the differences observed in these two cases. For this purpose, a direct comparison of the cooled and the un-cooled cases is performed via Figure 6.39 to Figure 6.42.

In each figure, the top plot coded as “A” gives either heat flux or Stanton Number distributions at each span for each profile under investigation. “R” and “U” in the entries describe the profile type as radial and uniform, respectively. The second group of plots coded as “B” gives the baseline that the deviations are calculated from. This is either the average of the temperature profiles, or the span-wise average within a given profile, depending on the type of the variation of interest. When it is the average of the profiles, the measurements corresponding to the same wetted distance locations from each profile are directly averaged, and the range bars show the variation between the two data points used (one data point per profile). For span-wise averaging, all available measurements from different spans, and some very close data points (when available) are averaged.
together. This calculation is performed separately for the two profiles. Therefore, the baselines and the following calculations in groups “C” and “D” are not inter-related between the profiles, but they are rather presented together. For instance, in Figure 6.33 group B, the measurements at around -35% wetted distance of group A (the plots at the top coded as “A”) at the same span are averaged over the two profiles (the solid and the empty red circle), whereas the orange shapes right next to the red circles are averaged together, separately. The cooled case refers to the nominally cooled case, and the range bars shown in group A give the variation between the nominally cooled runs used in the calculation of those data points. In the averages of the two profiles, the contribution of all these runs can be observed in the range bars. On the other hand, in Figure 6.34, the red and the orange shapes symbolizing 5% and 90% span are averaged together, but this averaging is done separately for the empty shapes (uniform profile) and the solid shapes (radial profile). Consequently, the range bars shown in group B for span-wise averaging gives the range between the spans averaged.
Figure 6.33 Variations Between Profiles on the Vane Airfoil Surface For Cooled Case
Figure 6.34 Variations Between Spans on the Vane Airfoil Surface For Cooled Case
Figure 6.35 Variations Between Profiles (Left) and Between Spans (Right) on the Vane Airfoil Surface For Cooled Case
Group C is the calculated deviation of each data point from the average at the corresponding location. Group D converts this deviation into a percent variation by normalizing the deviation amount by the average value at that location. The percent variations would be magnified if the average Stanton Number calculated were a near-zero point. This situation will arise in some upcoming plots. Generally, the percent variation scales are in the level of 75% (shown as 0.75 in the $-y$ axis), however it should be kept in mind that the actual values this variation refers to are indeed small Stanton Numbers. As was mentioned before, due to these small levels, the differences in heat transfer between varying inlet conditions (profiles or cooling levels) were not easily observable at some locations. Via working at percent variation level, these differences can be observed.

The resolution in data is shown with dashed lines in Figure 6.33 on the deviation plots of group C. It was stated in Section 5.2 that the typical resolution in Stanton Numbers is $\pm3\times10^{-5}$, and it is $\pm4000\text{W/m}^2$ for heat-flux measurements. Although not all calculated differences shown in group C are resolvable with the measurement accuracy, there is still a good amount of measurements outside of that resolution band.

Group A is repeated between Figure 6.33 and Figure 6.34, as this group presents the data in terms of heat flux and Stanton Numbers as a start point for the calculation. Stanton Number levels are smaller roughly by $1\times10^{8}$ times than the heat flux values, and looking at the absolute values would result in a misleading conclusion such as there is observable collapse by converting heat flux to Stanton Number. The variation between spans is observable when data is presented in terms of heat flux. But, this difference looks like diminishing once the conversion to Stanton Numbers is performed. However, a good measure for the amount of collapse occurring is observed via the comparison of groups D of heat flux and Stanton Number figures. The percent variation of group D reveals that there is some slight collapse of the data with conversion to Stanton Numbers for both the temperature profile differences and span-wise variation. This slight reduction indicates that there are possibly some other variation sources that are not accounted for in Stanton Number normalization. For example, the reference temperature was chosen as the local inlet temperature, disregarding the differences in the coolant temperatures. If it was possible to use any closer mainstream temperature measurements instead of the ones
collected at the upstream rakes for calculating Stanton Numbers (such as thermocouples on the leading edge), it is more likely that the span-wise distributions would collapse even further. In percent variations, it is not only the magnitudes reducing slightly, but the distributions of the data points are changing as well.

Figure 6.35 compares the variations caused by the profiles and across the spans in terms of Stanton Numbers. Apparently, at the same wetted distance, there is a larger variation across the spans than between the profiles. Of these locations, the leading edge is the largest variation observed. 5% and 90% spans deviate by largest amounts from the average for the radial profile. This is not surprising as there are temperature gradients introduced at the inlet with this profile.

When all spans are considered, the average difference between the uniform and radial heat transfer reduces down to 0.0001. Despite this low level of difference between the two profiles, Figure 6.35 shows that there are clearly observable differences between the profiles, and between the spans.
Figure 6.36 Variations Between Profiles on the Vane Airfoil Surface For Un-Cooled Case
Figure 6.37 Variations Between Spans on the Vane Airfoil Surface For Un-Cooled Case
Figure 6.38 Variations Between Profiles (Left) and Between Spans (Right) on the Vane Airfoil Surface For Un-Cooled Case
When there is no cooling, the heat transfer and Stanton Numbers shown in group A appear to be at much higher levels. Not only for the cooled case, but especially for the un-cooled case as presented in Figure 6.36, 90% span location mostly has higher heat flux than the rest of the spans, specifically at the front part of the airfoil. The uniform profile shows the most significant increase at 90% span when there is no cooling, but the radial profile shows elevated heat transfer levels as well. Despite the low inlet temperature in this region for radial profiles, the fluid still stays warmer causing higher heat transfer. In fact, the gradients at the outer span introduced at the inlet are not as strong as at the inner span. In addition, there could be an effect of a possible radial migration occurring towards the tip. Besides, the horseshoe vortex structure created at the leading edge causes elevated heat transfer levels, when combined with the endwall flow field effects. A similar flow structure was observed by Tallman et al. [69].

There is also some collapse of the raw data observed when Stanton Numbers are used for the un-cooled case. The largest reduction occurs at 5% and 15% spans in Figure 6.36 group D. A similar reduction is also observable for the cooled case of Figure 6.34. Figure 6.37 demonstrates the span-wise variations existing in the un-cooled environment. When there is no cooling flow in the environment, there is less variation introduced in the boundary conditions. This results in a better collapse of the data, which is mainly observed across the spans. Figure 6.38 group D shows that the variations between the profiles and the spans are at comparable levels.

Figure 6.39 and Figure 6.40 make a comparison of the cooled and the un-cooled cases for all four groups – A, B, C, and D. The differences between different spans and different profiles are clearly visible due to large level of Stanton Numbers in the un-cooled environment. Figure 6.40 also shows through the span-wise averages (group B) that the differences between the profiles are also much bigger in terms of Stanton Numbers (and in terms of heat flux as was shown in Figure 6.37). However, the percent variations from the averages are at comparable levels between the two cases, which means that the cooled case has much smaller heat transfer levels.
Figure 6.39 Variations Between Profiles on the Vane Airfoil Surface For Cooled (Left) and Un-Cooled (Right) Cases
Figure 6.40 Variations Between Spans on the Vane Airfoil Surface For Cooled (Left) and Un-Cooled (Right) Cases
Figure 6.41 Variations Between Profiles on the Vane Airfoil Surface For Cooled and Un-Cooled Cases – Percent Deviations
Figure 6.42 Variations Between Spans on the Vane Airfoil Surface For Cooled and Un-Cooled Cases – Percent Deviations
Figure 6.41 and Figure 6.42 focus on group D of these figures, for profile and span-wise variations respectively, while providing the associated heat flux variations. The cooling holes locations on the spans are shown with black rectangles on the x-axis. The degree to which the data collapses is very similar for the profile variations between the cooled and the un-cooled cases in Figure 6.41, but there is an observable collapse for the span-wise variations of the un-cooled data compared to the cooled case. This more pronounced collapse shows that most of the variation at the inlet is due to the inlet bulk temperature for the un-cooled case, and use of the Stanton Numbers to present the results remove it. Also, the similar percent variation levels between the cooled and the un-cooled cases show that even if the uniform and radial profile effects look very similar in heat transfer levels when cooled, there are still as significant variations as in the un-cooled case. It is only a matter of resolution capabilities to measure these variations.

In addition, Figure 6.41 also shows that the uniform profile always causes higher heat transfer than the radial profile when un-cooled, as group D gives positive deviation from the average for the uniform profile, and the negative for the radial at all locations. This dominance of the uniform profile is not as clear when cooled.

So far, the data on the airfoil surface shows that despite much lower heat transfer levels, the cooled case generally shows very similar variations in heat transfer to the un-cooled case, both due to a change in the temperature profile and across the spans. More specifically, across the spans, the variation decreases when there is no cooling in the environment. For both cases, there are slight decreases in the variations once switched from heat flux to Stanton Numbers, as the inlet variation sources are partially accounted for in the normalization. For no cooling, a larger collapse in Stanton Numbers was observed especially in the span-wise direction, since there is less variation introduced at the inlet. As future work, this data analysis could be supported with a statistical model as was performed for the inlet temperature profiles. Modeling would help in establishing empirical relationships between the data, working with multiple variables and not just along the wetted distance.
6.2.2 Temperature Profile Effects at the Inner Endwall

One of the most interesting regions in the vane section is the endwall region. According to the endwall flow field description of Langston [3], the flow field is highly three-dimensional creating vortical structures, such as horseshoe and passage vortices due to the splitting of the main inlet flow occurring at the leading edge. This secondary flow, which acts differently than the regular mainstream flow, has a governing role on both the endwall and the immediate nearby airfoil surface. A preceding work of Langston et al. [4] showed that this three-dimensional secondary flow is responsible for a decrease in the pressure differences between the pressure and suction surfaces. As much as this secondary flow affects the aerodynamic loading and turbine efficiency, it also has important consequences on turbine heat transfer. The data presented here is unique, as this is the first time some measurements at the endwall of a “fully-cooled” vane in the rotating environment are presented in detail, bringing some insight to flow physics in this region.

In this section, the variations in the inner endwall heat transfer that arise due to the variation in the inlet temperature profiles will be studied. In a similar approach to that of the airfoil surface, the analysis will be presented in groups of figures describing different conditions (i.e. cooled and un-cooled). Figure 6.44 to Figure 6.46 provide the analysis for the profile effects at the nominal cooling condition. Following the cooled case, the un-cooled case effects are examined in Figure 6.47 to Figure 6.49. The comparison of the profile effects in both cases will be given in Figure 6.50 to Figure 6.52. Since the endwall is a special section of the vane due to its two-dimensional contour shape that is loaded with a dense cooling configuration, contour plots are used in conjunction to the line plots to visualize the measurement locations with respect to the cooling hole locations. Figure 6.53 and Figure 6.54 will provide these contours for the two cases, both in heat flux and in Stanton Numbers. The actual location of the endwall cooling holes and gauges are shown in Figure 6.43.
The layout used in the data analysis for the inner endwall is split into three different geometric locations: endwall passage, endwall inlet, and endwall exit. In all the upcoming figures, these three locations are presented separately, and each have three different types of plots coded as A, B, and C. Group A demonstrates the source data, that is the uniform and the radial profiles for this analysis, as well as the average of the two profiles that will be used as the base to calculate the deviations with respect to. Group B shows the deviation of each temperature profile data set from this average. Finally, Group C gives the percent variation, which is the normalized deviation with the average profile.

The gauge locations in Figure 6.43 are shown with an accompanying numbering system for each part of the endwall. The figures in the analysis should be viewed with this information to appreciate the effect of gauge location on heat transfer. The inlet gauges are all on the left of the vertical leading edge (LE) line. The gauges are numbered in an increasing order in the positive pitch direction (+y axis). There are two gauges at position 6 (from different vane sectors). Half of the gauges are located upstream of the cooling holes as indicated in the figure. The endwall passage gauges are the gauges lying between the leading edge (LE) and the trailing edge (TE) lines. Gauge positions are numbered in an increasing order starting from the leading edge in the positive percent axial chord direction (+x axis). The endwall exit gauges are located to the right of the trailing edge (TE) line. The numbering system is such that the position numbers increase in the negative pitch direction (-y axis). There are two gauges at position 4 (also from different vane sections). Only one of the gauges from this group stays above the leading edge (LE) line.

The inlet gauge position labels are shown in Figure 6.44 for each heat flux data point. There is no measurement available at position 4 for the specific case shown (nominally cooling with uniform and radial profiles). Leading edge corresponds to 0% pitch location. The other leading edge location at 100% pitch is the leading edge of the neighboring airfoil above. Also given is the data resolution range on heat flux and Stanton Number deviations in group B. At the inner endwall, the repeatability for heat flux is $\pm 5000$ W/m$^2$, and it is $\pm 8 \times 10^{-5}$ for Stanton Numbers. Generally the variation between the profiles is much higher than this resolution band.
There is a slight reduction in the variations as the raw data is normalized and converted to Stanton Numbers. This variation between the profiles is on the level of 75%, and is similar to the variations observed on the airfoil surface. For the cooled cases, the Stanton Numbers do not take into account the effect the coolant temperatures have on reducing the driving temperature, which is another source of variation.

Both gauges at position 6 provided measurements in Figure 6.44. These gauges are downstream of some cooling holes, while the gauge at position 5 is upstream of those holes. One of the measurements obtained at position 6 is almost the same as the one at position 5. The appearance that there is no cooling effect for one gauge, but a higher heat-flux value for another gauge located on different airfoils shows the importance of maintaining circumferential uniformity and exact duplication of the position of the gauges relative to the cooling holes. A more significant increase in heat flux is observed at position 2, which is also downstream of the cooling holes, but between them (not in a direct line). The gauge at position 1 located upstream, for example, gives lower heat flux, indicating that the addition of cooling off an angle from the heat-flux gauge can lead to complicated vortex structures capable of significantly changing the surface heat flux.

Stanton Numbers for gauges upstream and downstream of the first row of cooling holes are calculated differently. For most measurements presented in this thesis, Stanton Numbers use the total of the core mass flow and the vane inner cooling. For the gauges located upstream of the first cooling hole row, only the core mass flow is used since they do not see any coolant. The calculations show that this results in 7% increase in Stanton Number for the upstream gauges, which is not resolvable. When using Stanton Numbers, the measurements from the two gauges at position 6 collapse to a single value, demonstrating that the differences in the conditions that the two airfoils were exposed to were removed. But, there is still a slight increase in Stanton Numbers of the gauges downstream of the holes. This shows that the migration of coolant in the endwall region is a complex flow structure, and not all the cooling holes provide coolant effectively. This was shown in the study of Kost and Mullaert [29]. The strong gradients caused by secondary flows could be creating an in-flow through some of the ineffective cooling holes, increasing heat transfer.
Figure 6.44 Variations Between Profiles at the Endwall Inlet For Cooled Case
Figure 6.45 Variations Between Profiles at the Endwall Passage For Cooled Case
Figure 6.46 Variations Between Profiles at the Endwall Exit For Cooled Case
Figure 6.45 and Figure 6.46 show the same detail for the endwall passage and the endwall exit locations. The percent variation between the profiles is reduced only slightly after normalization. Once again this shows that the main differences between the cases are not due to mass flow or the local driving temperature. One known source mentioned earlier is the effect the coolant temperatures have on the driving temperature, which is not being modeled with this definition of the Stanton Number. It should be noted that the plots of group C show a 75% variation from the average condition, but since the average values are low, it is important to look at the plots in group A and B before making any claims about whether the percentage change is significant or not. However, this variation is about the same on the airfoil surfaces.

In Figure 6.45, the heat transfer trend throughout the endwall passage shown in group A demonstrates more of a linear characteristic until around 60% axial chord. After this point, there is a notable rise in heat transfer, and it fluctuates beyond this location. This points to the acceleration of the flow at position 5, and the disturbance created in the flow results in alteration of the fluid temperatures as an increase or a decrease further downstream. Positions 4 and 5 are very close to the sonic line as could be inferred from Figure 6.43, and the addition of the cooling on top of the passage cross-flow makes the flow features more complicated here. The trailing edge region is exposed to the rotation effects of the downstream rotor, which contributes to the alterations observed beyond the sonic line.

Figure 6.47, Figure 6.48, and Figure 6.49 show the analysis at the endwall regions for the un-cooled case. In none of the locations do the variations of group C seem to be noticeably decreasing. This suggests that it is not only the coolant temperatures, but some other source of variation that contributes to the overall variation observed in the heat-flux. Since the resolution (shown in the plots) comes from repeatability measures, this variation is due to changes that are occurring among the cases being studied (i.e. the effect is real).
Figure 6.47 Variations Between Profiles at the Endwall Inlet For Un-Cooled Case
Figure 6.48 Variations Between Profiles at the Endwall Passage For Un-Cooled Case
Figure 6.49 Variations Between Profiles at the Endwall Exit For Un-Cooled Case
Next three figures, Figure 6.50, Figure 6.51, and Figure 6.52, present a comparison between the Stanton Numbers of the cooled and the un-cooled cases that were given so far. On the airfoil surfaces, it was observed that the percent variations caused by the profiles were at similar levels between the cooled and the un-cooled cases, but there was a significant difference in the heat transfer levels between the two. At the endwall inlet, it is not only the variations, but also the heat transfer levels becoming comparable between the two cases. However, the absence of the coolant raises heat flux and Stanton Numbers through the passage and towards the endwall exit, especially for the uniform profile. The data suggests an increase in the differences between the two profiles through the passage and at the endwall for no cooling. That is, addition of cooling decreases the differences between the profiles to some extent that are otherwise large when not cooled. This increase is due to the significant increase in the heat transfer associated with the uniform profile at these locations, but the percent variations remain at the same levels.
Figure 6.50 Variations Between Profiles at the Endwall Inlet For Cooled (Top) and Un-Cooled (Bottom) Cases
Figure 6.51 Variations Between Profiles at the Endwall Passage For Cooled (Left) and Un-Cooled (Right) Cases
Figure 6.52 Variations Between Profiles at the Endwall Exit For Cooled (Top) and Un-Cooled (Bottom) Cases
These observations made in the line plots are reflected in two-dimensional contour plots of Figure 6.53 and Figure 6.54, in terms of heat flux and Stanton Numbers, respectively. All available measurements from different airfoils are combined to produce this overall view of one single passage. Wherever there is no data available, the missing points are filled via interpolation. Therefore, in order to avoid a biased comparison, each case uses the same gauges. At the inlet, both profiles have similar heat transfer. A more homogeneous flow field is observed for the radial cooled case. There is a gradual increase through the passage for all contour plots shown. These gradients are much stronger for the uniform profile. With the acceleration of the flow towards the exit, there is more heat convected, resulting in higher Stanton Numbers. The thinning of the boundary layer towards the trailing edge also plays a role in the heating. These features are kept at a lower level for the radial profile. Since the Stanton Number removes the variations at the inlet temperatures using the closest rake’s inner span measurement, the originally existing temperature gradient at the inlet is not reflected in the distributions. Therefore, the contours in terms of Stanton Number reflect flow physics and the differences between the two profiles better. The larger difference at the endwall exit between the un-cooled profile cases observed in the line plots is visible via the sharp differences between the colors. The trends are assumed to repeat through the whole vane section. For example, the green spot at the trailing edge of the airfoil in the radial profile cooled case of Figure 6.54 can be assumed to be showing up at the same spot of each passage. But, the contours were not adjusted to show this repetition in the demonstration.

In the endwall region the amount of percent variation in heat transfer due to the profile effects is comparable to that observed on the airfoil surface for both the cooled and the un-cooled cases. The endwall region has a complicated flow field due to multiple effects occurring simultaneously, such as the existence of shocks (as this is a transonic turbine) and the rotation of the downstream rotor, all in addition to the secondary flow effects. As a result, the fluid temperatures fluctuate through the passage aft of the sonic line. When there is no cooling, the heat flux, especially for the uniform profile, has increased through the passage and at the endwall exit. However, the percent variations between the two profiles remain in the same levels throughout the endwall.
Figure 6.53 Heat Flux Contours at the Endwall For Cooled (Top) and Un-Cooled (Bottom) Cases
Figure 6.54 Stanton Number Contours at the Endwall For Cooled (Top) and Un-Cooled (Bottom) Cases
6.2.3 Hot Streak Alignment Effect

In the previous section, the variations between the uniform and radial profiles were quantified. These two profiles are circumferentially uniform, which means that the circumferential variation is small, as shown in Section 5.2. The third temperature profile discussed in this section is the hot streaks. Hot streaks constitute a different type of profile due to the fact that they vary in the circumferential direction by design.

In this experiment, every other airfoil was aligned with hot streaks, which is what would likely be seen in modern engine designs. This resulted in a ±8.5% circumferential variation for the hot streaks, larger than the uniform and radial profile variations (±1.8% and ±3.9%, respectively). In older designs, there is a larger gap between the aligned airfoils (for example every three airfoils) increasing the circumferential variation even further. As mentioned before, there are different alignment possibilities provided by activating different heater rods, with potentially different magnitudes. Due to the way the heater matrix was constructed, the upstream rakes were not exactly lined up with the hot streak generation locations. Therefore, the inlet temperature measurements did not measure the actual magnitude of the hot streaks, but rather one of the rakes always provided the hot streak magnitude at the 37.5% corresponding pitch location on the heater sector. Consequently, the fluid temperature measurements from this rake were used in generating Stanton Numbers to avoid introducing any changes coming from moving the rake relative to the hot streak.

The actual magnitudes of the hot streaks are measured at the heater sector and are provided via the centerline metal temperature measurements in the heater core. Both of these centerline measurements and the upstream rake measurements at the rake aligned with the 37.5% pitch of the heater sector are given in Figure 6.55. The magnitudes are presented in terms of metal temperature profile factor, previously defined as:

\[
\text{Profile factor} = \frac{T_{\text{local}} - T_{\text{ave}}}{T_{\text{ave}}} \tag{19}
\]
$T_{\text{ave}}$ is different than $T_{\text{avg}}$ that has been used so far in that it is the area-weighted average temperature instead of an arithmetic average, and $T_{\text{local}}$ is the local temperature. This definition removes both the average temperature and the shape variations.

Figure 6.55 Hot Streak Heater Sector and Rake Temperature Measurements

There are three different magnitudes for each alignment type. The left portion of Figure 6.55 shows the temperature distributions for the mid-passage alignment (MP), and the right shows the distributions for the vane leading edge alignment (VLE). The heater sector metal temperatures and the rake fluid temperatures are in agreement. The axial distance from the heater to the rake and the 35.7% pitch circumferential distance result in the largest deviations between the two measurements in the vicinity of the peak locations. The hot streaks are classified according to the magnitudes at the peak locations. According to this, Run 37 has the highest magnitude for the vane leading edge alignment, and Run 35 gives the highest magnitude for the mid-passage alignment. The temperature gradients observed for each class follows the same order with the magnitudes at the peaks.
In this section, the focus will be on the effects of the alignment (vane leading edge or mid passage) on heat transfer. The runs that will be studied will be of high and medium magnitudes, since the profile shape of the low magnitude runs are different. The regions of interest on the airfoils will be the mid-span suction surface and the inner enwall regions in this discussion. The corresponding inlet temperature measurements on the rakes are at the 10% and 50% spans. As is shown in Figure 6.56, the temperature measurements for the two types of alignments match very well for both magnitudes at all locations, other than the vicinity of the peak (around 60% span). For medium magnitude hot streaks, the mismatch extends to mid-span. This means that a direct difference of the
heat transfer levels will directly give the effect of alignment type on the heat transfer, other than the mid-span for the runs with medium magnitude, which should be kept in mind when examining the data.

In all figures presented in Section 6.1, the two alignment types were introduced as separate entries. Figure 6.11 showed that the general trend is a slight reduction at all spans when the hot streak is aligned with the vane leading edge, but this decrease is mostly within the variation between the runs in the same profile, suggesting that heat transfer level is generally insensitive to alignment type. Figure 6.57 and Figure 6.58 analyze the mid-span suction surface location for high and medium magnitude hot streaks. The solid shapes represent the alignment with the mid-passage, and the empty shapes represent the alignment with the vane leading edge. These figures show that there are more significant differences observed at mid-span on the suction surface: There is a decrease when the alignment is established in line with the leading edge for high magnitude, and one of the two available data points for medium magnitude case also suggests an observable decrease. This observable reduction is not some production of Stanton Number normalization, but is also seen in the raw heat-flux data. The similar trends between the Stanton Number and heat flux distributions suggest that even if there is a slight difference in the inlet temperature profiles for the medium magnitude hot streaks, this difference is not significant when the temperature profile reaches the vane surface.
Figure 6.57 Effect of Hot Streak Alignment Types With High Magnitude at Mid-Span Suction Surface

Figure 6.58 Effect of Hot Streak Alignment Types With Medium Magnitude at Mid-Span Suction Surface
The gauge distribution on the vane airfoils and the alignment type effective on each airfoil was shown in Figure 4.13. The data shown here comes from the instrumentation that are distributed over three different airfoils. Airfoil 17 receiving the leading edge alignment with the hot streak provides only the first data point on the suction surface that is closest to the leading edge. All the other measurements in the most heavily loaded region of the suction surface are split between Airfoils 16 and 35. The vane leading edge alignment data points were all measured on Airfoil 35, but the mid-passage alignment measurements come from both of the airfoils. Figure 4.13 indicates that when the hot streaks are aligned with the mid-passage, the pressure surface of Airfoil 16 is exposed to the hot streak, while Airfoil 35 is skipped. When it is switched to the vane leading edge alignment, the leading edge of Airfoil 35 is under direct effect and Airfoil 16 is skipped. However, measurements from both airfoils suggest similar trends independent of the alignment type. This is probably a result of the relatively denser distribution of hot streaks, i.e. alignment with every other airfoil; hence the differences observed by different airfoils have evened out. In an engine where there is one hot streak per a vane section of 4-5 airfoils, the influence of the alignment would expected to be more pronounced on the airfoil that is under direct contact with the hot streak compared to its surroundings. In more modern turbines with a combustor having this dense hot streak distribution, these differences turn out to be negligible.

In their rotating rig facility, Povey et al. [53] collected data on a high-pressure turbine vane in un-cooled environment, and observed that the pressure surface was insensitive to the alignment type. The findings here confirm those results. They also observed that the alignment with the leading edge increased the heat transfer on the suction surface, while an alignment with the mid-passage decreased it. They suggested that the increase for the leading edge alignment was due to the circulation occurring at the leading edge that drags the hot flow over the suction surface. Similarly, when the alignment was with the mid-passage, it was the colder flow being dragged over the suction surface via circulation.

The findings of this thesis show opposite trends to what is discussed above. But, the measurements presented here are not confined to 50% span only. Even though the
two alignment cases are very close to each other everywhere else other than 50% span suction surface as was shown in Figure 6.11, the average values still suggest a slight consistent decrease in Stanton Number for the alignment with the vane leading edge at all spans. One possible explanation to this difference in the findings is the effect of coolant in the environment, as this is the main known difference between the two experimental setups. Unfortunately, it is not possible to make such a comparison solely depending on the results from current research, since there is only one un-cooled hot streak run that is aligned with the mid-passage. Hence, the results from two different researches at different cooling conditions (no cooling versus nominal cooling) will be compared. The addition of the coolant to the vane flow field must be causing a change in the way the hot streaks interact with the airfoil surfaces. A passage vortex in the vane passage flow field, as defined by Langston [3], is responsible for carrying fluid from the pressure surface of an airfoil over to the suction surface of the neighboring airfoil. When there is coolant in the environment, it is dragged over with this passage vortex onto the suction surface of the neighboring airfoil. This coolant washes out the hot flow carried over to the suction surface via circulation occurring at the leading edge that is suggested by [53]. Therefore, lower temperatures are observed on the suction surface when the hot streak is aligned with the leading edge. On the other hand, when it is aligned with the mid-passage, the passage vortex that carries the coolant from the neighboring airfoil’s pressure surface interacts with the hot flow path created by the hot streak on its way through the passage. This results in an immediate increase in the temperature of the coolant that is being carried over to the suction surface. The coolant comes to a higher temperature by the time it arrives at the suction surface, causing an elevated heat transfer here.
Figure 6.59 Effect of Hot Streak Alignment Types With Medium Magnitude at the Inner Endwall
Figure 6.59 gives the heat flux and Stanton number distributions at the inner endwall for the medium magnitude hot streak with both alignment types. Reduction through the passage when the leading edge is aligned with the hot streaks becomes much clear in terms of Stanton Numbers. On the other hand, there is an opposite trend observed at the endwall exit in terms of a significant increase with vane leading edge alignment. The features observed at different sections of the endwall once again give clues about the complexity of the flow field in this region. First of all this is a transonic turbine, and the shock creation will augment heat transfer levels. Also, the rotation of the downstream rotor will contribute in additional unsteadiness at the endwall exit. As Langston [3] suggested and as confirmed by many other cascade experiments, the endwall secondary flows complicate mainstream flow. In addition to all these, in a cooled environment, the gradual addition of coolant through the passage adds more to this complexity.

The endwall passage has measurements obtained on Airfoils 12 and 13. According to Figure 4.13, Airfoil 12 sees a mid-passage alignment directly, and Airfoil 13 sees a leading edge alignment when the switch is made between the two. But, there is a consistent decrease with vane leading edge independent of the airfoil. In other words, the aligned and un-aligned airfoils receive similar effects from the upstream alignment condition.

Figure 6.60 shows the heat flux distributions at the inner endwall for the hot streaks of high magnitude. The Stanton Number distributions are not given, since Run 35 and Run 37 are among the runs having high uncertainty due to low driving temperatures. For high magnitude, there is no observable difference between the alignment types. Povey et al. [53] had also observed that the hot streak alignment type did not result in a change in favor of one or the other. This means that the magnitude of hot streak plays a role in the effect of alignment types.
In this section, hot streaks are analyzed separately than the other two profiles due to the different nature of their profile shape. The focus was on the alignment type effect on the heat transfer, and there were observable differences at specific locations throughout the vane. Both the heat flux and Stanton Number trends suggest a consistent decrease at mid-span suction surface, while there is still some slight decrease with vane leading edge alignment on the other parts of the airfoil surface as well. The complexity of the endwall flow field altered the effect of the alignment type in this region. While there
was no effect observed for the higher magnitude of hot streak anywhere at the endwall, for the medium magnitude hot streak, first a reduction was observed through the passage, and later it switched to an increase at the endwall exit.

6.3 Cooling Level Effects on Vane Heat Transfer

The turbine vane airfoils used for these experiments have a complex cooling configuration using hundreds of cooling holes via multiple rows through two separate cooling circuits: the vane inner and outer circuits. In addition to the vane cooling circuits, there is also a purge circuit that cools the disk area between the vane and the rotor. The mass flow can be controlled separately in each of these circuits, which made it possible to generate different cooling conditions. There are mainly four cooling levels: high, nominal, low, and no cooling. According to Table 5.2, the overall supplied coolant amount is about 10%, 13%, and 15% of the total mass flow for low, nominal, and high cooling levels, respectively, accounting for all the runs of the experimental matrix. Via turning the purge cavity and vane outer cavity on and off and changing the mass flows through these channels, additional subset of cooling conditions could be performed.

In the first two sections, the variations between the main cooling cases will be investigated on the airfoil surface and at the inner endwall. For the investigation of the cooling level effect, it is important to use the same temperature profile so that the observations made will be clearly due to the change in the cooling level. The runs with radial profile were performed with the largest set of cooling conditions, which is why the analysis is based on these runs given in Table 6.1. The low cooling level used a group of runs that have different purge flow amounts. Purge flow was found to have an impact on the heat transfer of the downstream rotor of this turbine stage confined to the platform area [83], however, it was not found to have an influence on the vane (in the upstream direction). This is shown via a comparison of different purge flow rates in the last part of Section 6.3. The effect of the vane outer cooling variation is shown in that same section. For the nominal cooling level, the repeat runs with and without vane outer cooling are compared.
6.3.1 Cooling Level Effects on Airfoil Surface

In the analysis, both heat flux and Stanton Number distributions are provided. The variations between cooling levels (Figure 6.61), and the variations among spans occurring for each cooling level (Figure 6.62) will be analyzed in a comparative manner, in Figure 6.63 and Figure 6.64. The span-wise average of each cooling level is calculated independently of the other cooling levels. In addition, there are two different baselines used in this analysis: the nominal cooling and the average of cooling levels. The figures mentioned above use the nominal cooling as the baseline. The average of cooling levels is used in Figure 6.65, and the comparisons are performed in Figure 6.66 and Figure 6.67.

When using the nominal cooling case as a baseline, the effect of the un-cooled case can be observed, but it is difficult to observe the individual differences between the other cooling levels. However, these differences become more observable when using the average of the cooling levels as a baseline. Even if calculations show there are differences among cooling levels, the reduction obtained at individual spans is not distinguishable, but this effect becomes clear when all spans are viewed together, as will be demonstrated in NSR plots (Figure 6.68 and Figure 6.69).

A similar analysis technique to the one applied when observing the temperature profile effects will be used for the cooling level effects. Each figure consists of four different groups coded as A, B, C, and D. Group A presents the data, while group B gives the average or the baseline data that will be used in the calculation of the deviations. Groups C and D give these deviations in terms of absolute and percent values.

The entries used in the data presentation are “no” for the un-cooled case, “low” for low cooling, “nom” for nominal, and “high” for high cooling levels. The spans follow the same color-coding of the previous section.

Figure 6.61 group A shows the heat flux and Stanton Number distributions for all cooling levels. Here, the variations of each cooling level from the nominal condition are given. The resolutions in heat flux data and in Stanton Number are shown with dashed lines.

The percentage variations are important in comparing different cases (cooling levels here) with each other, but whether these variations are significant or not can be
determined only by looking at the absolute variations. Even for the percent variations of 100%, the absolute differences in Stanton Numbers are in the level of only 0.0001-0.0003. The absolute difference for each cooling case from the un-cooled case is consistently in an average of 0.0004 Stanton Number over all spans, while the difference between the low and high cooling levels is only around 0.0001.

The percent variations are magnified when the baseline data point used in the normalization is very close to zero. An example is observed at around -20% wetted distance. All cooling levels give much higher variations at this location, since the nominal cooling measurement is almost zero. Therefore, these data points are the artifacts of the normalization. As a side note, Group D shows that there is really no collapse of the data when normalized.

Figure 6.62 shows the span-wise variations for each cooling level. One observable result is that there is much less variation across the spans when there is no cooling. There is again no collapse of the data observed when converting to Stanton Numbers, but the distributions have changed compared to the heat flux distributions. That is, normalization takes care of some of the variation sources such as driving temperatures, but not all are removed. Figure 6.63 gives a comparison of all four sections between the cooling level and span-wise variations in terms of Stanton Numbers. The un-cooled case had much higher deviation from the nominal cooling case as expected.
Figure 6.61 Variations Between Cooling Levels on the Vane Airfoil Surface
Figure 6.62 Variations Between Spans on the Vane Airfoil Surface
Figure 6.63 Variations Between Cooling Levels (Left) and Between Spans (Right) on the Vane Airfoil Surface
Figure 6.64 shows the comparison between the cooling levels and the spans in a zoomed version. The top row shows the variations on the heat flux side, and the bottom is for the Stanton Numbers. On the left, the variations for the cooling levels are given. The scales are adjusted at 2 (i.e. 200%), taking out the portion of the un-cooled data beyond those limits.

The solid circles represent the un-cooled case. The variation levels are more clearly observed at this scale. The un-cooled case gives a much smaller variation across the spans compared to all the other cooling levels. This was observed earlier when the un-cooled case was examined for the temperature profile effects. This finding suggests that addition of cooling aggravates the heat transfer gradients across the spans.

The differences from the nominal cooling show a tighter range on the suction surface compared to the pressure surface. That is, the differences between the cooling levels are much clearer on pressure surface. The deviation range is dictated by high cooling level on this surface for both cooling variation and span-wise variation; and it is dictated by low cooling on the suction surface for both variations. The largest variation is observed with high cooling level, and this takes place at 5% span in terms of a decrease from the nominal cooling. With the limited data from each span, it is hard to make a comparison of the coolant behavior between the pressure and suction surfaces. According to the passage flow field model proposed by Langston [3], the passage vortex should be sweeping the coolant away within itself from the pressure surface. Despite this phenomenon, high cooling level reduces the heat transfer level at 5% span on the pressure surface by about 100% more than the nominal cooling. The level of the percent variation is caused by the relatively low value of the Stanton Number for the nominal cooling case. But the significance of this number is more apparent when compared to the other cooling levels at 5% span, which are substantially less, indicating that this is an area where the increase in cooling has a large effect.

This comparison of the cooling variation to the span-wise variation given in Figure 6.64 suggests a slightly larger variation among the spans, especially on the suction surface. This can be better appreciated when the un-cooled data points on the left are not taken into account. The calculations for the variation between cooling levels are based on the nominal cooling, and the no cooling case is expected to be much higher than the
nominal case. Span-wise variations look slightly larger, and this may be because of the
flow field features such as secondary flows. For example, 90% span variations from
the average for different cooling levels generally are higher. There is a noticeable increase at
the leading edge area on suction surface. This is immediately downstream of the
showerhead. On the other hand, the cooling flow variations are purely driven by the
differences in the coolant mass flow rates. Because the cooling variation is calculated
with respect to the nominal cooling case, there are fewer data points available (i.e. no
nominal cooling) in comparing the left with the right side.

Instead of using nominal cooling, calculating the deviations from the average of
the three cooling levels (low, nominal, high) provides more data points as shown in
Figure 6.65. Figure 6.66 shows the variations with respect to this cooling average and
with respect to the nominal cooling. The nominal cooling, by the supplied amount of
coolant, is roughly half way between the other two cooling levels, and therefore, both
calculations suggest similar trends to that from the average. The similarity between these
two cases means that the variation observed is not random. With the addition of the
nominal cooling entry, the trends have become much clearer. For example, the cooling
variation has become much tighter.

As a last step in this comparative analysis, cooling variation with respect to the
average can be studied in conjunction with span-wise variation, as it provides a more
filled-in distribution. In Figure 6.67, there is now a much clearer collapse of the cooling
levels on suction surface.
Figure 6.64 Variations Between Cooling Levels (Left) and Between Spans (Right) on the Vane Airfoil Surface – Percent Deviations
Figure 6.65 Variations Between Cooling Levels With Respect to Average of Cooling Levels on the Vane Airfoil Surface
Figure 6.66 Variations Between Cooling Levels With Respect to Average of Cooling Levels (Left) and With Respect to Nominal Cooling (Right) on the Vane Airfoil Surface – Percent Deviations
Figure 6.67 Variations Between Cooling Levels With Respect to Average of Cooling Levels (Left) and Between Spans (Right) on the Vane Airfoil Surface – Percent Deviations
One parameter commonly used to quantify the film-cooling performance was defined by Sen et al. [8] as the Net Heat Flux Reduction (NHFR) introduced with equation (4), and is used to quantify the reduction in the heat flux due to the coolant effect.

\[
\text{NHFR} = \Delta q_r = \frac{q_r^0 - q_r^*}{q_r^0} = 1 - \frac{q_r^*}{q_r^0} = 1 - \frac{h_r (T_{aw} - T_w)}{h_0 (T_{aw} - T_w)}
\]  

Due to the nature of the coolant ejection process, the heat transfer coefficient at the ejection site increases, while the addition of the coolant works in the opposite direction. NHFR helps quantify the net change in the heat transfer coefficient in terms of the resulting net heat flux. Similarly, to quantify the reduction in Stanton Numbers, Net Stanton Reduction (NSR) is defined per equation (11).

\[
\text{NSR} = 1 - \frac{\text{St}_r}{\text{St}_0}
\]  

In the analysis to be presented here, a slightly modified version of this parameter as given by equation (12) accounts for the effect of the negative Stanton Numbers in the calculation.

\[
\text{NSR} = \frac{\text{St}_0 - \text{St}_r}{|\text{St}_0|}
\]  

Additionally, the hole diameters may vary in the direction of the flow, resulting in geometry-dependence in the observations made. In order to isolate the cooling effects only, the distance downstream of the holes are normalized using the hole diameter, which has been commonly used in flat-plate experiments. The idea is that cooling provided by a hole with larger diameter should be more effective on the downstream location than a hole with smaller diameter, and using a x/d scale enables one to track the influence of coolant downstream from the hole, regardless of the hole diameter.
For the analysis, the 50% span suction surface is chosen, since it is heavily loaded with heat-flux gauges that will provide a clear trend of cooling behavior. This surface provides most of the data among all spans, giving the highest resolution in the heat transfer distribution. There are five rows of cooling holes on this surface starting from the leading edge. The first four rows extend to 16% wetted distance. The last cooling row is at 35% wetted distance. The majority of the gauges are located downstream of this last cooling hole.

Figure 6.68 Stanton Number and Net Stanton Reduction at Mid-Span Suction Surface

Figure 6.68 shows the mid-span suction surface data from different perspectives. The first plot at the top shows the data in actual wetted distance locations with respect to
The leading edge. Beyond the dashed lines are the measurements downstream of the last cooling hole. The middle plot is the NSR distributions with respect to x/d, with the location of each gauge being calculated from the last hole. The NSR definition given by equation (12) used the un-cooled case as the base Stanton Number, defined as St₀. It is known that the un-cooled case is not an accurate representative of an actual un-cooled airfoil with no cooling configuration on the suction surface. For this reason, NSR is also defined with nominal cooling as the base parameter, St₀, and is called a Relative NSR. This distribution is shown in the bottom plot of Figure 6.68.

The goal is to detect the effect of different blowing ratios on effectiveness downstream of the cooling hole. In film cooling literature, the blowing ratio is commonly used to quantify the coolant amount. With the experimental setup and instrumentation for the current research, the overall coolant through each circuit is measured, and different coolant blowing ratios are obtained by varying the mass flow rate. Then, the blowing ratios through individual holes are assumed to be evenly shared. Each cooling level shown here therefore represents a different blowing ratio. Net Stanton Reduction parameter can be treated as the effectiveness, since both are used to quantify the coolant influence on reducing heat transfer levels.

Figure 6.68 shows a similarity between the high and nominal blowing ratios. The position x/d= 2 is the closest gauge location to the upstream hole. Although, the differences between the cooling levels are not very clear given the large range bars the low cooling case has, there are still generally two different trends observed. The low blowing ratio has the highest Net Stanton Reduction (NSR) value immediately downstream of the cooling hole. It gradually decreases further downstream. The second trend is observed with the nominal/high blowing ratios. The trend indicates an initially low reduction in heat transfer right downstream of the hole. The reduction in Stanton Number, or the effectiveness for that matter, first gradually decreases until x/d=9, then starts increasing up to x/d=16. The clear difference between the low cooling and the nominal/high cooling cases suggests that things are happening in the vicinity of x/d=16. Although small, the difference observed between the two trends is that the second group of blowing ratios could be beyond the optimum value that causes a lift-off in the region immediately downstream of the hole. The increase in the net reduction occurs with the
reattachment of the coolant jet back on the surface. This reduction starts diminishing again further downstream. The resemblance of the two higher cooling levels has also become more visible with the relative NSR plot at the bottom of Figure 6.68. High cooling level is mostly very close to zero when NSR is calculated with respect to the nominal cooling. Low cooling gives less reduction. The most significant difference between the two cooling levels that occurs at x/d=16 location is also clear in the relative NSR plot.

The trends observed here points out to the typical behavior of the coolant on a convex surface. Ito et al. [25] made similar observations in his work when using a gas turbine blade in a cascade. Although there are five rows of holes on the suction surface, the last row is sufficiently downstream of the previous row (by 19% wetted distance) that it very much shows similar trends to that of a single row of holes, which is what Ito et al. [25] studied.

Other than the clear difference observed at x/d=16, all other measurements are within the range bars, which is an indication of not as clear differences between cooling levels. This is what was observed in Figure 6.67 as well. Most of the mid-span measurements are performed over a distance where there are no cooling holes. It may be that the combined cooling effect of the heavily distributed scheme of cooling holes is overcoming the differences between the cooling levels on the airfoil surfaces, which is within 5% of the overall mass flow. If the range between these cooling levels were much larger, clearly the cooling amount effect could be distinguished even at the regions with a higher number of holes.

The largest x/d range was available for the mid-span suction surface. However, this is not so on the other parts of the airfoil surface, due to the gauge locations with respect to the cooling holes. The NSR trends at each span will be presented with respect to the leading edge in terms of wetted distance as usual; along with the cooling hole locations, with Figure 6.69.
Figure 6.69 Net Stanton Reduction on the Vane Airfoil Surface

All NSR calculations are based on the un-cooled measurements. At 5% span pressure surface, the effect of the high cooling case is observed the most clearly. This reduction in heat transfer is in agreement with what was observed in Figure 6.64 via large negative deviations for high cooling from the nominal (or the average). Other than this span, generally the differences between the cooling levels are not very clear. One other interesting observation is that there is a consistent decrease in NSR values from near-hub
(5% span) towards the tip. This is observed for all cooling levels. That is, the heat transfer is increasing, and the highest level is reached at the tip, which was also observed before.

So far, the cooling level effects on the airfoil surface have been discussed. The distributions showing the percent variations of each cooling level from the baseline (the baseline is the nominal cooling, average of cooling levels, or span-wise average of an individual cooling level, depending on the type of the calculation) were found to be changing slightly after converting the heat-flux measurements to Stanton Numbers, but no further collapse of the data was observed, suggesting that not all the variation sources were removed via normalization. Less variation between the spans was apparent for the no-cooling case. For the other cooling levels, the span-wise variation was found to be slightly larger when compared to the variation between the cooling levels. The differences between the cooling levels are more observable on the pressure surface, especially with the high cooling level at 5% span. The NSR trends showed the typical effect of the convex surface observed on the coolant behavior. The highest reduction in heat transfer is achieved at the inner span, and decreases towards the tip.

6.3.2 Cooling Level Effects at the Inner Endwall

The cooling level effects for the inner endwall are shown in Figure 6.70, Figure 6.71, and Figure 6.72, for the inlet, passage, and the exit of the inner endwall, respectively. Figure 6.43 can be used to visualize the measurements with respect to their locations at the endwall. The endwall line plots will be followed with the contour plots of Figure 6.73 and Figure 6.74 to provide a two-dimensional view of the flow field. Finally, as the cooling effects are examined in this section, with the NSR trends given via the contour plots of Figure 6.75.

Figure 6.70 shows that varying the coolant amount does not result in sufficient differences in heat transfer to be resolved in the measurements. The majority of the data points lie within the resolution bars. Further, the un-cooled measurements are all observable. In percent variations, the measurement at around 33% pitch location has nominal cooling measurements that are very close to zero, which is responsible for the magnified percent variations as shown in group C. There is no collapse of the data when
converting from heat flux to Stanton Numbers. This is not only true for the inlet, but at all sections of the endwall.

The observations that were made at the endwall passage when examining the profile variations in Figure 6.45 are also observed in Figure 6.71. The data points shown by green circles representing the nominal cooling case are the data points used in representation of the radial profile in the analysis of temperature profile effects. The measurement at around 55% wetted distance is also magnified when normalized with the nominal cooling due to the low levels, and should not be focused on. The data points at positions 4 and 5 are very close to the sonic line, which may be the reason to why there is a sudden rise at position 5. These two gauges, referring to Figure 6.43, are very close to and in line with each other, while being very close to cooling holes at the same time. The main difference from what was observed at the inlet is that the percent variations between the cooling levels (group C) have increased through the passage towards the exit. The fluctuations lead to a ±250% variation from the nominal cooling. At the exit, these percent variations have decreased to the level observed at the inlet, as shown in Figure 6.72. The deviations (group B) show that the heat transfer levels throughout the endwall region are of similar levels, although slightly increasing at the endwall exit. Thus, the change in percent variations is due to the extremely low nominal values. The deviations on the endwall are also of comparable level to those found on the airfoil surface.
Figure 6.70 Variations Between Cooling Levels at the Endwall Inlet
Figure 6.71 Variations Between Cooling Levels at the Endwall Passage
Figure 6.72 Variations Between Cooling Levels at the Endwall Exit
Once again, the contour plots serve to visualize the endwall cooling configuration. The minimum values in the contour plots do not get as low as what is shown in the scales, and are actually higher at least by two color bars than the lower-end limits. In Figure 6.73 and Figure 6.74, the heat flux and Stanton Number distributions for the four cooling conditions are given respectively. In terms of heat flux, not much of a difference can be observed. The differences start showing up in Stanton Numbers. Other than the un-cooled case, there is no visible difference between the cooling levels at the inlet. The un-cooled case shows more difference with much higher Stanton Numbers. The difference between the other three cooling levels start showing up through the passage.

For the no-cooling case, the two extremes of the range are observable at various locations. There is a hot spot at the front of the leading edge of the airfoil, and half of the endwall inlet region on the side of the pressure surface is about twice as hot as the other portion extending towards the suction surface. However, one would expect to have a more blended trend at the endwall inlet if the contours were to be extended over the next airfoil passage to obtain continuity. The pressure and suction surface lines look significantly different. While there is heat transfer from the wall to the gas path by the pressure surface as indicated by the blue region, this is reversed on the suction surface. This is most likely a result of the passage vortex. There is the horseshoe vortex acting on the suction surface, which contributes to the increase in the heat transfer level on this surface.

Even low amounts of cooling (around 10% of the overall mass flow) is enough to smooth out the sharp differences observed without cooling. The hot spot that was observed at the middle section of the exit is pushed to the aft of the trailing edge region. The nominal and high cooling levels show no clear significant difference. With both cooling levels, clearly the temperature differences are blended out further, resulting in more homogeneous heat transfer distributions. The secondary flow existence may be partially erased by the addition of cooling to the environment, starting from the un-cooled case gradually towards the high cooling case.
Figure 6.73 Heat Flux Contours at the Endwall For Different Cooling Levels With Radial Profile
Figure 6.74 Stanton Number Contours at the Endwall For Different Cooling Levels With Radial Profile
In addition, depending on the coolant jet lift-off phenomenon occurrence, the surface heat transfer will be adversely affected immediately downstream of the cooling holes. It is interesting that the trends of the nominal and high cooling levels were found to be very similar in the Net Stanton Reduction trends of the 50% span suction surface as well, as was shown in Figure 6.68. According to the trends of 50% span, it was suggested that the nominal and high cooling cases might have blowing ratios beyond the optimum point that possibly caused jet lift-off of the surface. This may have some contribution to why the two cooling levels affect heat transfer at the endwall in a very similar fashion. The behavior of the coolant beyond the ejection point will vary depending on the endwall contouring, as it has been shown by many studies (such as [25]) that the effectiveness of the coolant differs between suction (convex) and pressure (concave) surfaces, and therefore is very much surface curvature dependant.

At this point, it will be useful to look at the NSR distributions in the endwall region. Figure 6.75 shows the contour plots of NSR values of the three different cooling levels. As was done for mid-span suction surface, the relative NSR values are also calculated for the low and high cooling levels with respect to the nominal cooling level. These two plots are shown at the second row of Figure 6.75. NSR scale is shown in the range between 1 and –1 for all cooling levels, and the relative NSR range is shown between 2 and –2. The range for relative NSR is larger since the calculation is based on the nominal cooling case that has lower heat flux values compared to the un-cooled case. There are a few points that are out of these ranges given, hence they have the colors of the extremes. Positive values show a reduction in Stanton Number, whereas negative values show an increase. There are fewer data points presented, since in order to calculate NSR, Stanton Numbers from two cases are needed at the same time: 1) the case under investigation, and 2) the un-cooled case (nominal cooling for the relative NSR calculation), which decreases the number of available data points calculated.
Figure 6.75 Net Stanton Reduction Contours at the Inner Endwall For Cooling Levels
For the low cooling level, there are both high reductions and high increases in Stanton Number. The high reduction occurs generally at the inlet and front part of the endwall passage, and also near the suction surface. The high increases occur near the pressure surface. This behavior can be explained using the endwall model of Langston [3]. This is the result of passage vortex acting across the vane passage flow field, which is responsible for carrying fluid from the pressure surface of one airfoil over to the suction surface of another. When there is coolant in the environment, it is dragged over with this passage vortex onto the suction surface of the neighboring airfoil. This causes reduction in Stanton Number on the suction surface, and an increase on the pressure surface.

NSR trends also look very similar for the nominal and high cooling levels just like the Stanton Number distributions did, which is expected. However, it is interesting that the low cooling case does not have only larger areas of reduction and increase in Stanton Number, but also the reduction and increase amounts are also higher compared to these two higher cooling cases. That is, the increase in cooling does not necessarily decrease the Stanton Number even further, but rather acts in a way to even out the differences in the flow field. All these result in a clear difference between the two groups of cooling levels – low and nominal/high – at the endwall, compared to the airfoil surfaces.

Regarding the relative NSR contours, a large reduction is observed for the low cooling level through the passage relative to the nominal cooling level. The data showed this trend in the line plots of Figure 6.71. For high cooling level, the majority of the flow field is colored with green, which is an equivalent of an NSR value of 0. This indicates that other than a few spots, the high cooling level very much resembles the nominal cooling level at the inner endwall.

The data at the inner endwall suggest that adding cooling to this region makes the flow more complicated. The deviations in heat transfer caused by cooling variation were similar to what was observed on the airfoil surface. There was a slight increase in deviations at the endwall exit when the entire endwall section is considered. The contour plots were functional in terms of visualizing the locations that receive the highest amount of cooling, which became even much clear via the NSR trends in this region. When there is no cooling, the passage vortex showed its effect by creating large differences in heat transfer level across the region.
With the addition of cooling, the temperature differences were blended out, but the difference between the low cooling and nominal/high cooling was much clearly observable when presented in terms of NSR. Low cooling case gives both the highest and the lowest reductions in heat transfer, along the suction and partially along the pressure surfaces, respectively. This indicates that the coolant is transferred from the pressure towards the suction surface within the passage vortex. The supply of additional cooling has results specific to the location at the endwall in a way to smooth out the differences in the flow field, rather than providing a consistent decrease in heat transfer everywhere.

### 6.3.3 Purge and Vane Outer Coolant Flow Effects

After looking at the major effects of coolant amount, the effects of minor changes in cooling levels, such as purge flow and the vane outer cooling addition should be examined. In this section, first the inlet conditions will be presented via the inlet temperature profiles. The purge flow effects are demonstrated in Figure 6.77 and Figure 6.78, on the airfoil surface and at the endwall, respectively. The vane outer effects on the airfoil surface and at the endwall will be shown in Figure 6.79 and Figure 6.80. As there are some clear effects observed with the addition of vane outer coolant, a comparative study is performed between this cooling and the other cooling levels that were analyzed before. These comparisons will be presented in Figure 6.81, Figure 6.82, and Figure 6.83, in terms of deviations and percent deviations.

According to Table 5.1, the runs that have the variable purge flow and variable vane outer flow are the ones with radial inlet temperature profile. Therefore, this section will concentrate on the comparisons for radial inlet temperature profile only. Runs 18 and 19 do not have purge flow. Run 25 has a medium level purge, while Runs 17, 23, and 24 have high purge cooling. All of these have low vane inner and vane outer cooling flows, and they provide a good study case to investigate the influence of purge flow on the vane heat transfer. Similarly, Runs 27 and 28 have vane outer cooling, while Runs 40 and 41 do not. These four runs are all nominal cooling runs, therefore the influence of vane outer cooling should be reasonably isolated. These runs are listed in Table 6.2.
Table 6.2 Runs Presented For Purge and Vane Outer Coolant Effects

<table>
<thead>
<tr>
<th>PURGE CONDITION</th>
<th>NO PURGE</th>
<th>NOMINAL PURGE</th>
<th>HIGH PURGE</th>
</tr>
</thead>
<tbody>
<tr>
<td>RUNS</td>
<td>18,19</td>
<td>25</td>
<td>17,23,24</td>
</tr>
<tr>
<td>VANE OUTER CONDITION</td>
<td>NO VANE OUTER</td>
<td>WITH VANE OUTER</td>
<td></td>
</tr>
<tr>
<td>RUNS</td>
<td>40,41</td>
<td>27,28</td>
<td></td>
</tr>
</tbody>
</table>

Figure 5.2 and Figure 5.3 had showed that there are significant variations between different temperature profiles, but the profile shapes for each cooling condition with the same profile were found to be very similar, while having a very tight circumferential variation. The circumferential variation for different levels of purge cooling and the vane outer cooling are illustrated in Figure 6.76. The group of runs with all different purge flow levels, and the runs with different vane outer flow levels are shown in separate plots to illustrate these two study cases. The first row of plots shows the actual profile shape for each condition. Although it looks like there are slight differences in profiles between each condition, these differences are removed when each profile is normalized with the average rake temperature, as shown in the second row of plots. This means that these variations are due to the inlet bulk temperature differences between conditions. Each data point in the last row of plots shows the average of normalized temperatures from repeat runs at the closest rake, and the error bars show the overall peak-to-peak variation between the runs at that rake and at the corresponding span. There was no significant distinction observed between the profile shapes of all cooling levels in Figure 5.3. Since the runs investigated in this section make up a subset of the low cooling and nominal cooling cases of Figure 5.3, the distinctions between these subset runs are still within the variations across all cooling levels, as is expected. The result is that the profile shape effect is considered to be isolated for these runs.
Purge flow is introduced through the cavity between the vane and the downstream rotor disk. The purge is a small amount of coolant flow, less than 1% of the total turbine mass flow, and is used to keep the wheel-space cavity cooler by preventing the hot gas ingestion from the mainstream. At the same time, it aids in cooling the leading edge region of the blades. With purge flow introduction, the static pressure at the exit of the wheel-space cavity opening to the mainstream between the two discs increases, causing a blockage at the vane exit.

The interest in the open literature has been mainly concerning the influence on the aerodynamic performance caused by this addition of coolant amount – even if small – for the
wheel-space cavity cooling. Pau et al. [79] observed that the purge flow was responsible for enhancing the rotor endwall secondary flow. The cooling influence was mostly limited to the platform. This is also what is observed on the downstream rotor of the current research program [83]. Both the heat transfer and the temperatures of the fluid outside of the boundary layer on the platform showed an influence of purge flow, regardless of the amount. Purge flow caused a reduction in both heat transfer and local temperatures. Pau et al. [79] suggested that the blockage effect of the purge introduction on the vane exit flow field helped with reducing the trailing edge shock losses by forcing the shock to move backwards in the vane section.

It will be interesting to see if this aerodynamic phenomenon has any impact on the upstream vane heat transfer distributions at all. As the rotor studies suggest, the cooling effect of the purge flow is most likely to be confined to the inner span and the endwall region. The distributions presented in Figure 6.77 suggest that the purge flow does not have an influence on the vane airfoil surface at any of the span locations. The most likely location, 5% span, is not affected by purge flow. Figure 6.78 shows the same comparison performed at the endwall. The influence of the purge flow is not distinguishable in this region, either. There are large variations both at the inlet and the exit. However, there is no significant influence observed on the heat transfer distributions neither on the airfoil, nor at the inner endwall.
Figure 6.77 Heat Flux and Stanton Number Distributions on the Vane Airfoil Surface With Purge Flow Variation
Figure 6.78 Heat Flux and Stanton Number Distributions at the Inner Endwall With Purge Flow Variation
There are two cooling circuits used to supply coolant to the airfoil surfaces. The vane outer cooling supplies coolant gas mainly to the trailing edge slots and the outer endwall region. Hence, vane outer coolant can be interpreted as trailing edge cooling during the analysis. The last three rows on the pressure surface also take their share from this coolant. According to Table 5.2, roughly a 5% of the overall mass flow was allocated to supply coolant to these regions.

As the aerodynamic shape of the airfoil is critical to prevent performance losses, the trailing edge is designed to have a very thin structure, which makes it one of the airfoil sections that is highly sensitive to high temperatures. This usually brings trade-off between the aerodynamic performance and thermal stresses, necessitating a sufficient cooling scheme be implemented at the trailing edge to prevent significant reductions in turbine airfoil life. To reduce performance penalties associated with unnecessary coolant supplies to this region, the design of the trailing edge cooling scheme becomes an important issue to tackle. There have been numerous studies specializing on the trailing edge cooling design.

Roback and Dring [41], while analyzing the hot streak impact on the downstream rotor, also were able to determine locations where the coolant accumulated. The upstream vane coolant was found to be collecting on the suction surface, even if the cooling slots were located on the pressure surface. The experiment of Abhari and Epstein [77] had a comparative cooling study of vane trailing edge injection. The Nusselt Number for the leading edge of the downstream blade was found to be reduced by 18%. There was a monotonic increase on the pressure surface, but almost no change was observed on the suction surface with the addition of the trailing edge cooling. These were the two studies of the many that investigated the influence of trailing edge cooling on the downstream blade row. The more recent experiment performed for the downstream blade row of Build 2 [83] revealed similar results. The vane outer had significant contribution in reducing the heat transfer of the airfoil, with the coolant having more impact on the suction surface. Dunn [72] studied the influence of trailing edge cooling on the mid-span vane heat transfer, in addition to the influence of the presence of the rotor. The portion of the suction surface downstream of the ejection site did not show any effects on the heat transfer due to the cooling. The Stanton Numbers on the pressure surface were reduced as the blowing ratio was increased.
In the analysis performed in this section, there are two groups of runs examined according to Table 6.2. The measurements from each group represent the average of two runs, and the bars show the range within these runs. The runs with vane outer coolant are the same runs that were previously examined as nominally cooled radial runs. These are runs 27 and 28. The group with no vane outer coolant consists of Runs 40 and 41.

The airfoil surface distributions in Figure 6.79 show that the Stanton Number significantly increases towards the trailing edge at 5% span when there is no vane outer cooling. This is not surprising, as the last three rows of holes on the pressure surface (blue arrows), supply additional cooling when the vane outer channel is turned on. The most significant increase is at 5% span on the pressure surface, by an average of 0.0002 Stanton Number, but a similar increase is generally observed at all spans towards the trailing edge. The average increase of all spans is in the level of 0.0001. The mid-span suction surface looks less responsive to the vane outer cooling. The location closest to the leading edge at 15% span has actually two groups of runs collapsing to the same value. The sparse data on the suction surfaces of the other spans do not tell too much about the general trends of the heat transfer here, but the single points closer to 50% wetted distance at 15% and 90% spans also suggest an increase.
Figure 6.79 Heat Flux and Stanton Number Distributions on the Vane Airfoil Surface With and Without Vane Outer Flow
Figure 6.80 Heat Flux and Stanton Number Distributions at the Inner Endwall With and Without Vane Outer Flow
Vane outer cooling is not confined to the trailing edge region and these holes only, but the outer endwall also receive cooling from this circuit. As a result, the effect is not limited to the pressure surface where the trailing edge coolant is supplied through, but the suction surfaces are also influenced. As an additional remark, the decrease in heat transfer due to the addition of the vane outer cooling was more significant on the upper half of the rotor airfoil surfaces [83]. On the vane, its influence can be observed at all spans more evenly.

Figure 6.80 presents an effective increase in Stanton Number by 0.0002 at the endwall exit when there is no vane outer cooling. At all other sections of the endwall, there are no significant differences between these two cases. A continuous increase in the fluid temperatures was observed through the passage without vane outer cooling. With the developing horseshoe vortices starting from the leading edge region, the circulation in the flow must be increasing the temperature of the flow field via convection. However, there is a simultaneous increase at the metal temperatures compensating for the increase seen in fluid temperatures, resulting in no significant change in heat flux.

The increase in heat transfer in the absence of the vane outer cooling is noticeable both on the airfoil surface as a consistent increase growing towards the trailing edge, and also at the endwall exit. To quantify the level of change in heat transfer produced by supplying vane outer cooling, the absolute and percent deviations of the cooling case with no vane outer from the nominal condition will be compared to those of low and high cooling cases.

Figure 6.81 shows the variations on the airfoil surface in heat flux and in Stanton Numbers. As stated before, the nominal cooling is indeed the cooling case with vane outer. Therefore, the deviation plots give the difference of “no vane outer” case from that with vane outer circuit on. The comparison suggests the largest positive deviation from the nominal cooling be achieved by “no vane outer” case. There is a substantial increase in heat transfer generally at all spans when there is no outer coolant supplied. This increase is consistent between absolute and percent deviation plots. This difference is at a comparable level to that of high cooling on 5% span pressure surface, and is also observable on suction surface.

Figure 6.82 shows the deviations of the cooling levels and the condition with no vane outer cooling from the nominal case, at the endwall. The deviations for the “no vane outer” case from nominal case has simply been added onto the previously presented deviations in
group B of Figure 6.70, Figure 6.71, and Figure 6.72. Similarly, Figure 6.83 gives the percent deviations of all cooling conditions in a comparative manner at the endwall. Other than the “no vane outer” case, all other data was presented in group C of the aforementioned figures. The two locations, at around 55% axial chord in the passage, and at around 33% pitch location, have the nominal cooling case measurement very close to zero, as was shown in Figure 6.70 and Figure 6.71, and are therefore magnified as an artifact of the percent calculation. These figures show that the vane outer coolant effect is comparable to the effect of other cooling levels at the endwall as well, but its effect does not stand out as much in terms of percent variations.

As a summary, the purge flow did not show an observable effect on the vane row, neither on the airfoil surface, nor at the endwall region. On the other hand, the effect of vane outer coolant supply was significant, especially towards the trailing edge on the airfoil surface, and at the endwall exit. The variation in heat transfer caused by having this circuit on is found to be comparable to the effect of other cooling levels on the pressure surface, and is also observable on the suction surface. The vane outer coolant has also similar effects to the other cooling levels at the endwall.
Figure 6.81 Effect of Vane Outer Cooling Compared to Other Cooling Levels on the Vane Airfoil Surface
Figure 6.82 Effect of Vane Outer Cooling Compared to Other Cooling Levels at the Inner Endwall – Deviations
Figure 6.83 Effect of Vane Outer Cooling Compared to Other Cooling Levels at the Inner Endwall – Percent Deviations
6.4 Comparison of Temperature Profile Effects With Cooling Level Effects

Up to this point, several comparisons have been presented between different conditions in order to quantify the variations these conditions produce in heat transfer. Knowing by how much and by which factors vane heat transfer is affected is critical in taking the necessary actions for avoiding overheat of the vane airfoils. In the analysis performed for the investigation of the temperature profile effects, it was observed that the variations caused by temperature profiles in a cooled environment were at very comparable levels to the variations occurring across the spans from hub to tip on the airfoil surface. The percent variations were found to be similar to those in an un-cooled environment, meaning that even if the Stanton Numbers are much smaller in the cooled environment, the differences between the profiles are still observable as long as the measurement resolution permits it. The span-wise variation was found to be even higher for the cooled case compared to that observed in the un-cooled case. At the endwall, the differences and the percent variations between the profiles were similar to those on the airfoil surface when cooled. When there was no cooling, the passage and the endwall exit showed similar variations, but larger differences in heat flux magnitudes occur between the two profiles because the uniform profile has a larger influence.

In the analysis performed regarding the cooling variation, the differences in heat transfer caused by varying the cooling amount were found to be at similar levels with the span-wise variation on the pressure surface, but these differences became smaller on the suction surface. At the inner endwall, the differences were similar to what was observed on the airfoil surface, with an observable increase at the endwall exit mimicking the profile effects in this region. As a result of low heat flux values obtained with cooling, there were significantly much higher percent variations found in a growing fashion through the passage compared to what was seen on the airfoil surface.

In this section, these variations observed in the two sections will be compared with each other. The cases to be compared will be the profiles at nominal cooling and the cooling levels with radial profile. The airfoil surface comparisons will be given in Figure 6.84 to Figure 6.87, and the endwall comparisons will be provided in Figure 6.88 to Figure 6.91.

Figure 6.84 and Figure 6.85 present the comparison of the cooling level and profile variation in terms of deviation and percent deviation, both with respect to the average case of
each. As was noted before, the measurement at around -22% wetted distance on the pressure surface in Figure 6.85 has the average values for both cases very close to zero, as group B of Figure 6.39 and Figure 6.65 suggest, and the percent deviations at this location are magnified. Disregarding this data point, the ranges of deviations and percent deviations for both parameters are similar. The larger variation on the pressure surface observed for cooling variation is mostly dictated by the significant heat transfer reduction achieved with high cooling.

Figure 6.86 and Figure 6.87 compare the span-wise variations for both parameters. As there are more runs included in the analysis of cooling effects, there are more data points. High cooling shows a large range between the spans producing elevated heat transfer at the tip, and the lowest heat transfer at 5% span. This trend was also observed in the NSR distributions of Figure 6.69 as a reduction in heat transfer growing from the tip towards the hub. These features are also observed for the profiles. Generally, the span-wise variations between the two parameters are also at similar levels.

Figure 6.89 to Figure 6.91 show a comparison of the variations observed at the endwall between the two parameters in terms of heat flux and Stanton Numbers. Due to the low heat flux value, the percent deviations for the data points at around 67% and 77% axial chord location in the passage are magnified. The variations between the profiles at the endwall are similar to those on the airfoil surface. The same observation can be made for the cooling level variation. There are much higher percent variations through the passage due to low heat flux values for cooling levels. However, the comparison between the actual magnitudes of the two parameters (Figure 6.89 and Figure 6.90) indicate that the differences at the endwall region are significantly higher for the profile effects compared to the cooling level effects. There is an increase in the differences at the endwall exit for both cases.
Figure 6.84 Comparison of Deviations Caused by Cooling Levels (With Radial Profile) and Profiles (at Nominal Cooling) on the Vane Airfoil Surface
Figure 6.85 Comparison of Percent Deviations Caused by Cooling Levels (With Radial Profile) and Profiles (at Nominal Cooling) on the Vane Airfoil Surface
Figure 6.86 Comparison of Span-Wise Deviations Caused by Cooling Levels (With Radial Profile) and Profiles (at Nominal Cooling) on the Vane Airfoil Surface
Figure 6.87 Comparison of Span-Wise Percent Deviations Caused by Cooling Levels (With Radial Profile) and Profiles (at Nominal Cooling) on the Vane Airfoil Surface
Figure 6.88 Comparison of Deviations in Heat Flux Caused by Cooling Levels (With Radial Profile) and Profiles (at Nominal Cooling) at the Inner Endwall
Figure 6.89 Comparison of Percent Deviations in Heat Flux Caused by Cooling Levels (With Radial Profile) and Profiles (at Nominal Cooling) at the Inner Endwall
Figure 6.90 Comparison of Deviations in Stanton Numbers Caused by Cooling Levels (With Radial Profile) and Profiles (at Nominal Cooling) at the Inner Endwall
Figure 6.91 Comparison of Percent Deviations in Stanton Numbers Caused by Cooling Levels (With Radial Profile) and Profiles (at Nominal Cooling) at the Inner Endwall
Overall, a change in the temperature profiles or the cooling amounts results in similar variations in heat transfer levels on the airfoil surface. The variations between spans for the two control variables (profile and cooling) are also similar. At the endwall region, although the percent variations caused by cooling variation are much higher through the passage due to low heat fluxes, the actual differences in heat transfer levels due to profiles are significantly higher than those due to cooling levels.
CHAPTER 7
CONCLUSION

As part of the Heat Transfer initiative of the University Strategic Alliance (USA) Program sponsored by General Electric Aviation (GEA), a full-scale stage and one-half high-pressure turbine designed and manufactured by GEA was instrumented, housed in a test rig, and multiple experiments were performed at the OSU Gas Turbine Laboratory (GTL). The Build 2 experimental program focused on obtaining the full heat transfer map through the first stage of a rotating turbine operating at design corrected conditions. What makes this particular research primarily unique is that the vane was fully cooled and measurements were obtained for many locations on the airfoil and endwall with different vane inlet temperature profiles. Purge flow was provided to the disk cavity and to the un-cooled (but fully instrumented) blade row. Another thesis [82] has addressed the experimental results for the blade row in detail. One additional thesis in final preparation addresses comparison of the purge flow characteristics with CFD prediction. The goal of this program was to obtain an extensive database, which will serve as the basic design information for more efficient film-cooled turbines. In order to achieve these designs, the importance of advancements in CFD design tools cannot be underestimated. These tools can only be improved if a reliable data set is available. The experimental setup of this research was tended to mimic actual engine operating conditions as closely as can be done under controlled laboratory conditions. It is important to establish a thorough understanding of the influence of all the studied parameters on heat transfer and cooling effectiveness.

The combustor exit temperature profile is one of the major factors influencing the first stage vane heat transfer. A second major factor influencing heat flux level is cooling gas magnitude. Both of these concepts, when incorporated with detailed measurements on an extensively film-cooled vane, are providing a database, which will enable designers
to have a more complete picture of what is going on during the actual engine operation, and therefore, is definitely a unique contribution to the current state-of-the-art.

As there are many parameters contributing to the overall heat transfer, in order to draw correct conclusions from the observations made, one needs to isolate the parameters of interest properly. Many of the conditions that would occur during actual engine operation environment were successfully reproduced during the experiments, enabling several comparisons that were useful to gain insight into engine operation. The uncertainty analysis showed that the uncertainty caused by the error propagation arising from factors such as instrumentation accuracy, calibration and processing techniques was much lower than the variation among the repeat inlet conditions. On the other hand, this variation was small compared to the differences between different inlet conditions, clearly indicating that each operating point could be well resolved. A further implementation of a statistical model to the inlet temperature profiles gave predictions in good agreement with the experimental data within a tight 95% confidence limit. As the model made use of many runs, tight confidence limits could be obtained even for single runs. This statistical modeling could be used for the vane experimental data for the same purposes.

The data analysis for this thesis focused on the heat transfer levels on vane airfoil surfaces and endwalls. The influence of variation in inlet temperature profiles and the variation in cooling levels were investigated. Any possible effects of the purge and the vane outer flow on the heat transfer levels were questioned. A comparison between two different builds gave insight to how closely an uncooled airfoil environment could be replicated using an airfoil setup with an already implemented cooling scheme. All these efforts resulted in several key observations summarized below.

Addition of cooling resulted in notable reductions in heat transfer levels, but the calculations revealed that the percent variation in heat transfer caused by the temperature profiles are still comparable to that observed in an un-cooled environment. The variation at different spans caused by a profile shape also returned similar levels to the variation between the profiles, but this span-wise variation was reduced when there was no coolant supplied. That is, high temperature levels affected spans more proportionally without
cooling. On the other hand, the uniform profile showed dominancy over the radial profile mainly at the endwall exit when there is no cooling aid, resulting in larger differences, but similar percent variations between the two profiles in this region. The temperature gradient of the radial profile, and the complex flow nature of the endwall created by the secondary flow features all contribute to the differences observed between the two profiles.

The heat transfer distributions revealed general aerodynamic flow characteristics such as acceleration in the flow, boundary layer disturbances, and shock location as an increase in heat flux. The profile migration at the tip, and the secondary flow effects at the inner endwall and the tip were all observable in an environment that has the combined effect of cooling flow. The endwall flow characteristics of the un-cooled and the nominally cooled vane were further analyzed with a unique presentation of contour plots, underlying the large differences in heat transfer trends when the temperature profile is switched, and when the cooling is turned off. The heat transfer levels at the endwall were generally found to be parallel to those on the airfoil surfaces.

In a nominally cooled vane, the alignment of the hot streaks with the vane leading edge lowers the heat transfer compared to an alignment with mid-passage, both through the endwall passage and at mid-span suction surface. At the endwall exit, the vane leading edge alignment resulted in higher heat transfer. For a higher magnitude of hot streaks, however, the effect of alignment type was not observed at the endwall. The pressure surfaces were also found to be generally insensitive to the alignment type. In addition, it was observed that for a dense distribution of hot streaks, the aligned and un-aligned airfoils receive the same effects from the upstream alignment condition. This holds for all sections of the airfoil, including the endwalls.

The comparison between different cooling levels gave important information regarding how the coolant was distributed through the vane section. This information is useful for optimization of the coolant amount as knowing which parts of the vane need more coolant can assist in reconfiguring the cooling scheme implemented on the airfoil in a way to direct the coolant to the neediest parts. This will cut down the unnecessary coolant supply in favor of the overall engine performance. In data analysis, whenever the
endwall heat transfer was presented, contour plots were also used to give an idea for the two-dimensional behavior of the flow field in this region. The endwall contour plots showing the unique set of heat transfer data with and without cooling provided more insight to how the endwall secondary flow affects the local value of heat transfer.

Any amount of coolant addition to the un-cooled environment made a clear difference in the heat transfer, but the differences caused by different amounts of coolant were not as clear on the airfoil suction surface. Having a wider range of coolant supply could make these differences more observable. The clearest difference between the cooling levels was observed at 5% span on the pressure surface, making the span-wise variation on this surface at a defined cooling level similar to the variation between the cooling levels. The tip of the airfoil received higher heat transfer, while the inner span (5%) was more effectively cooled. Interestingly, the contour plots revealed that an increase in coolant did not necessarily increase coolant effectiveness at the inner endwall. The secondary flow structures causing sharp differences in heat transfer levels were overcome with an increase in coolant amount. The largest percent variations between the effects of cooling amounts in the vane section were observed at the endwall passage due to very low heat transfer levels taking place in this region with the addition of cooling. This resulted in cooling variation effects comparable to the profile variation effects when viewed in terms of percentage. The deviations, on the other hand, showed that the profile variation has more significant effect in the heat transfer levels in this region. The differences were elevated at the endwall exit by either varying the profiles or the cooling levels.

The secondary issues of interest were the possible influence of the purge flow and the vane outer coolant flow on the vane heat transfer. During the rotor heat transfer analysis of the same turbine stage, the vane outer coolant was found to have a clear effect on the blade heat flux, whereas the influence of the purge flow was confined mainly to the platform area [83]. The analysis presented in this research revealed that there was no influence of purge flow on the vane heat transfer, neither at the inner endwall, nor on the airfoil surface, as expected. The vane outer cooling showed an overall decrease in heat transfer levels at all spans that becomes more significant towards the trailing edge. This
effect was confined only to the exit region at the endwall. The reduction achieved in heat transfer with addition of vane outer cooling is at a nearly equivalent amount by itself to the reduction obtained by supplying the highest level of cooling.

A comparison between Build 1 and the un-cooled Build 2 vane heat transfer data was performed. The leading edge region and the pressure surface heat transfer measurements were found to be in agreement between the two builds, but the suction surface was found to give lower heat transfer suggesting ingestion on the pressure surface and ejection on the suction surface. A corresponding comparison for the inner endwall illustrated good agreement between the two data sets suggesting that ingestion on the endwall is not a significant effect. Supplying a small amount of coolant just enough to avoid the ingestion would help achieve mimicking the engine environment for an un-cooled case in the presence of a cooling configuration.

Additionally, it was observed that the driving temperature was the main driver in the Stanton Number uncertainty, influencing the endwall and 5% span locations for a few runs only. This finding suggests that to ensure reliable Stanton Number normalization, the temperature difference introduced by the inlet temperature profile should be adjusted in such a way that the temperatures at the inlet stays higher than the corresponding local fluid temperatures on the airfoil.

In the Appendix, Stanton Numbers calculated with different definitions were presented. Using local rake temperature measurements instead of the average inlet temperature was observed to result in a significant change in the calculated Stanton Numbers, especially for the high temperature gradient region in the inner endwall vicinity. The better collapse of the measurements in terms of Stanton Numbers suggests the use of local temperature measurements, since doing so removes the differences in the bulk temperatures and in the profile shapes. The effect of local temperature use was also clearly observed with extending the inlet temperature profile for a sample run via predictions at the inner endwall and 5% span locations. On the other hand, the change in Stanton Number caused by accounting for the coolant either partially (only vane inner circuit) or as a total (both vane inner and vane outer circuits) in the mass flow definition was not found to be observable.
Implications for Future Work

In this set of experiments, one span of instrumentation was mounted per airfoil. However, neither the vane section flow aerodynamics nor its heat transfer is circumferentially symmetric, and airfoils can be under the influence of slight differences. The cooling supply variations introduced at each channel to different airfoils also contribute to these differences. Locating the gauges assigned to different spans on the same airfoil could help avoid these variations if space permits. If not on the same airfoil, these same-span gauges could be located on the neighboring airfoils to keep a possible circumferential variation at a low level.

In a complex flow field as that of vane that is governed by the three-dimensional vortex structures at its inner and outer endwalls in addition to the temperature gradients created at the inlet, having more measurements will improve the resolution in the regions of interest. This will enable to capture more detailed flow physics, and bring more insight to the unknown flow structure. This requires more instrumentation. Increasing the number of the thermocouples per rake and distributing them to cover the inner endwall region, as well as increasing the number of rakes will result in an improved inlet temperature profile resolution. Locating another set of rakes right downstream of the vane trailing edge line will also assist in observing the influence of the vane on the distortion of the temperature profiles until they arrive at the downstream rotor. The vane could also be instrumented with thermocouples at the leading edge to track down the profile change from the upstream rakes to the vane section, or it could be more densely loaded with gauges to observe this change through the vane section. However, as this is a film-cooled vane, the complex cooling scheme did not leave enough room for this extra instrumentation.

In addition to instrumentation, increasing the number of repeat runs increases the certainty level of measurements proportionally. Similarly, an increase in the number of heated runs with similar conditions to those of un-heated ones will provide sufficient data to calculate the adiabatic wall temperatures.

There are some more improvements that could be incorporated to the future testing experiences. This research did not study the influence of inlet turbulence. It is
known from several cascade studies that the combustor exit turbulence levels have significant effects on heat transfer through the vane section. It is not only that, but the flow physics in the regions nearby to a cooling hole will be also altered depending on the free stream turbulence and coolant parameters such as blowing ratio. This, in turn, may increase or decrease the cooling effectiveness. Therefore, implementing a turbulence grid upstream of the cooled turbine in addition to the rotating rig environment and matched corrected parameters will help to more realistically reflect the actual turbine environment. As a result, one of the future plans is to install a turbulence grid to the experimental setup.

With all the lessons learned and observations made with the detailed data analysis in this research, important conclusions are drawn that will have an impact in future engine design and accompanied computational work. It should be noted that while as much as 20% of compressor air is used for cooling purposes in gas turbines, nearly half of this amount is used only for high-pressure stage vane cooling. This requires specific attention to the vane, since any improvement in film effectiveness in this region could be highly beneficial to the overall performance. However, a film-cooled vane requires hundreds of cooling holes to achieve full coverage, which turns it into a very complicated modeling job for the computational effort. Collecting measurements from this type of a loaded vane configuration both with cooling holes and the instrumentation is not any simpler task for the experimentalists, either. This makes the results presented in this thesis especially a valuable information source. Besides, the requirement of high inlet temperatures to obtain more power output is the driving mechanism today behind improving film-cooling technology that is applied in any sort of land-based engines for power generation or jet engines. That is, the problem in hand that needs to be solved is not specific to jet engines only, an example of which has been described for this study, but also the turbomachinery industry as a whole will benefit from the results of this research.
BIBLIOGRAPHY


261


266


This section explains possible Stanton Number definitions that might be used in a cooled experiment, discusses the advantages and disadvantages of each, and performs comparisons among these cases. As demonstrated, the Experimental Matrix has many different temperature profiles and cooling levels. There are also unintended variation sources for the inlet conditions introduced for each run, such as small variations in inlet bulk temperatures and the overall mass flow rates between repeat runs. All these variations were summarized in Table 5.2 and Table 5.4. The purpose of using Stanton Numbers is to remove the variation from the effects that are known to change the heat transfer level, so that the Stanton number variation between compared cases could be observed and properly ascribed to the changes in the experimental conditions. Using both Stanton Numbers and the basic heat-flux measurements (as done in data presentation) helps to isolate these effects. The definition of Stanton Number is repeated here one more time with equation (9):

\[
St = \frac{h}{\rho \cdot U \cdot C_p} = \frac{\dot{m}}{\dot{m}_\text{ref}} \left[ \frac{q''}{(C_p T)_{\text{ref}} - (C_p T)_w} \right]
\]

The possible variation sources accounted for with this normalization are the mass flow, the reference area, and the reference temperature. The other parameters used in the equation are determined by the condition of interest, i.e. the heat flux, and the wall temperature.

In all figures presented in Chapter 6, Stanton Number was defined by using the local temperature measurements from the closest rake to the instrumentation as the reference temperature. For hot streak runs, the local measurements from the rake that is aligned with the 37.5% pitch of the heater sector were used. The reference area was taken...
as the throat area, and the mass flow was defined as the total of the core flow and the vane inner coolant. As expected, with a large variety of experimental conditions, Stanton Number could be defined in various ways other than the approach used in this thesis.

One of the variables that could be modified in Stanton Number definition is the reference area. For an un-cooled turbine, the reference area on the vane section is usually taken to be the vane inlet area, but the throat or the exit choke areas could be utilized for the calculation as well. The key is that in the un-cooled case, these are not varying. When it comes to the cooling experiments, however, the addition of coolant causes a reduction in the available throat area for the mainstream flow. This is due to the blockage effect arising with the introduction of the coolant into the flow field. Therefore, the throat area is not constant anymore, but rather depends on the cooling conditions. This introduces multiple possibilities in terms of how to define Stanton Number. One could still use the vane inlet area and the core mass flow disregarding these minor changes through the airfoil passage. Another approach could be to use the throat area and the total amount of flow passing through it. This later approach is used through this thesis, and the rest of this section will focus primarily on the effects of changing the temperature and the mass flow parameters on Stanton Number.

Section 1 will handle the reference temperature effect, and the use of the average inlet rake temperature will be compared to the use of the local temperature. Section 2 will deal with the mass flow by comparing the effects of including different coolant streams in the calculations. Section 3 focuses on the use of local temperatures in Stanton Number definition on the inner endwall region consisting of the endwall and 5% span of the airfoil, and utilizes predicted values at 0% span and 5% span rake locations via an extended temperature profile. As will be remembered, the rakes did not have thermocouples at these locations, and therefore the 10% rake measurement was used to derive the endwall and 5% span Stanton Numbers. The results are that the temperature definition causes significant Stanton Number changes in the regions with high gradients, but the effect of coolant amount accounted for in the mass flow parameter is not observable.
1) **Stanton Numbers With Varying Reference Temperature Definition**

The variation in the inlet bulk temperature from run-to-run has an impact on the raw heat flux data, and it needs to be accounted for. It was noted before that two inlet rakes were used to perform the span-wise temperature measurements at equal portions of the turbine annulus area. Even if there are local temperature measurements available at these rakes at different spans, they are not expected to exactly reflect the corresponding local temperatures further downstream at the vane inlet. Clearly, the temperatures at both locations will not be identically the same due to the factors such as the contraction of the geometry from the vane inlet towards the vane throat area, the three-dimensionality of the flow nature, and the flow physics associated with the airfoil and endwall junction. All these contribute to the mixing of the flow field in the passage, which result in a reduction in temperature differences in the radial direction. The differences between the inlet temperature profiles do not contribute to this mixing, as was theoretically shown by Munk and Prim [116]. According to this, the mainstream flow will keep the same streamline pattern across the vane section as long as it is isentropic. Therefore, even if there is possible mixing in the flow due to some other factors, it can be assumed that each upstream rake location can be related to the corresponding downstream location on the airfoil within a streamline. As a result, the local temperature measurements from the upstream rakes are used, since that provides an indication of the profile effects, even if the measurement occurs upstream of the vane.

As an alternative, the average of all local measurements from both rakes could be used in the normalization. This measurement is the average inlet temperature (the inlet bulk temperature, \( T_{0,\text{inlet}} \)). Stanton Numbers using this temperature will be denoted as “Stanton Number (2)”. The original Stanton Numbers that used the local temperatures will stay as “Stanton Number”. Using the average temperature instead of the local temperature gives a less realistic estimate of the local driving temperature, but is probably a better-known value due to multiple measurements.

Figure A.1 uses the average inlet temperature to calculate Stanton Number (2) for the case of nominal cooling. The distributions given on the left are from Figure 6.11. This comparison between the two Stanton Number definitions shows that they yield close
results at higher spans although with observable differences, but these differences start showing up much clearly at 15% and 5% spans. As was explained before, if the upstream temperatures are very close to the metal surface temperatures, which occurs at the regions near the wall, the accuracy of the Stanton Number calculation deteriorates according to Figure 5.12. That is why some of the data points from those runs were excluded at 5% span of the left figure. When the average temperature is used in the normalization, the driving temperature is much larger at the inner spans; therefore, Stanton Number (2) definition has more points at 5% span.

Figure A.2 provides the Stanton Number distributions calculated using the local and average temperatures for the un-cooled radial run. The difference between the two cases is observable. There is almost no change from the heat-flux measurements when the average temperature is used, but the data points line up noticeably when the measurements are normalized by the local temperature. The reason is in that with using local temperatures, the temperature gradients are taken into account. This can be deduced by looking at a similar comparison for a uniform run of Figure A.3. By either using the local temperatures or the average temperatures, it does not seem like the Stanton Numbers change the trends observed in heat-flux measurements. Since uniform profiles have no gradient effects to be removed, using local temperature instead of average temperature does not result in a difference.
Figure A.1 Stanton Number Distributions Using Local Temperature (Left) and Average Temperature (Right)
Figure A.2 Effect of Using Local Temperature in Stanton Numbers for Radial Profile
Figure A.3 Effect of Using Local Temperature in Stanton Numbers for Uniform Profile
Figure A.4 shows the percent change in Stanton Numbers caused by using the average temperature in the normalization instead of the local temperature, and reflect the differences observed between the two temperature definitions in Figure A.2 and Figure A.3. This change is calculated as:

$$\%\text{Change} = \frac{St(T_{\text{avg}}) - St(T_{\text{local}})}{St(T_{\text{local}})} \times 100$$  \hspace{1cm} (31)$$

The results are presented by span location. For the radial profile, the consequence of a change in the temperature definition is large reductions in Stanton Number when using the average temperature of the rake at the inner spans. For mid-span and the tip, there is an increase. The amount of change is very similar for all the radial profiles. Uniform runs are affected less by the temperature. The trend observed over spans is similar to that of the radial, however, by a lessened amount. Among uniform runs of the experimental matrix, the un-cooled uniform run slightly differs. A consistent increase was observed at all spans of around 6%, other than the two points at 90% span, as shown
in Figure A.3. For a typical Stanton Number of 0.0003, this is a very small change. On the other hand, the reduction amounts of up to 60-70% for the radial runs are significant (around 0.0002) when typical Stanton Numbers in the cooled environment are considered. The large differences observed here shows that temperature definition is important for specific span locations. The better collapse of the data at those regions suggests that the use of the local temperature as the indicator of the driving temperature is the proper definition.

In summary, when the local temperatures are used instead of the bulk temperatures as the reference temperature in the normalization, there is not much change observed in Stanton Numbers for the uniform profiles, but the change observed for the radial profiles is significant. This is because by using the local temperatures, the temperature gradients are taken into account, which does not exist for the uniform profiles by nature. Therefore, using the local temperatures removes the differences in the bulk temperatures, as well as the differences in the profiles shapes, revealing the actual differences in the heat-flux measurements between the compared conditions.

2) Stanton Numbers With Varying Cooling Definition

Both the inlet and the exit mass flows in this experiment are well known via accurate measurements, and could be used in the normalization. However, the total flow going through the vane throat area is not well understood since the vane outer cooling circuit has parts that go through this area and parts that are injected into the flow aft of the choke area. As a result, it was considered that the vane inner coolant is injected over a large portion of the airfoil surface before the throat, and the contribution of the outer endwall cooling in the overall mass going through the throat area was ignored. This offered an approximate solution to the problem, which would otherwise require a very detailed procedure. Hence, Stanton Numbers presented in data analysis utilize the total of the inlet core flow and the vane inner coolant. As an alternative, the mass will be defined as the total of the inlet core flow, the vane inner coolant, and the vane outer coolant. That is, the total of the coolant mass supplied is accounted for in the mass flow, instead of partially considering the coolant effect. As will be remembered, the vane inner circuit supplies coolant to the inner endwall and the airfoil surfaces, and the vane outer circuit to
the outer endwall, the last three rows of holes on the pressure surface, and the trailing edge slots. The purge flow is introduced through the wheel space cavity, and was shown not to have an effect on the vane heat transfer, since it is supplied downstream of the throat area. As a result, this amount is not accounted for in the mass flow definition.

Keeping the temperature definition the same, i.e. using the local temperature measurements as in the original Stanton Number definition, the influence of the mass flow on the calculated Stanton Numbers can be observed by only varying the mass flow parameter. This new definition will be called “Stanton Number (3)”.

Figure A.5 presents the new Stanton Number distributions for comparison to those of Figure 6.19 with different cooling levels. This time, the differences between the two definitions of Stanton Numbers are not even visible. Compared to the much clear changes observed with a change in temperature definition, the changes caused by a change in mass flow definition are almost not observable. This is not surprising since the total mass flow has only increased by roughly 5% with the addition of vane outer coolant. On the other hand, the temperature gradients due to the inlet profile shapes can cause temperature differences up to 40K between neighboring span locations. This resulted in a more pronounced temperature influence in the normalization compared to the coolant effect.
Figure A.5 Stanton Number Distributions Using Local Temperature (Left) and Average Temperature (Right)
The statistics for one of the repeat runs constituting the nominal cooling case of Figure A.5 is given in Figure A.6 graphically. The calculation used equation (32):

\[
\text{%Change} = \frac{St(\text{Core} + Vi + Vo) - St(\text{Core} + Vi)}{St(\text{Core} + Vi)} \times 100
\]  

(32)

![Figure A.6 Effect of Coolant Mass in Stanton Numbers](image)

There is no difference between the spans. Adding the vane outer coolant mass flow to the overall mass flow used in normalizing the heat-flux measurements influences all measurements at every section of the airfoil similarly. All runs are found to follow this same behavior regardless of the temperature profile, as would be expected. The 5% reduction caused in Stanton Numbers is equivalent to \(1.5 \times 10^{-5}\) for a typical Stanton Number of 0.0003. This is even smaller than the resolution capability of the current instruments, which is \(\pm 3 \times 10^{-5}\). Therefore, addition of 5% more coolant in the normalization is not observable.
3) **Stanton Numbers With an Extended Temperature Profile**

The temperature profiles created by the heat exchanger at the turbine inlet were measured by thermocouples located at two upstream rakes in front of the first stage vane. These rakes were instrumented with miniature butt-welded thermocouples at different spans. The inlet temperature profiles presented throughout the data analysis showed these measurement locations. According to this layout, a region starting from 10% span by the inner wall up to 90% span by the outer wall was scanned. Since there are heat-flux gauges located at the inner endwall and 5% span locations on the airfoil surface, the closest inlet temperature measurements on the rakes were used for Stanton Number calculation for those gauges. In this section, the goal is to determine if Stanton Numbers would have changed significantly had there been measurements available at 0% and 5% span locations on the rakes. If it turns out to be that there is indeed a pronounced change in Stanton values, this may point out to the need of having higher measurement resolution closer to the endwall region at the rakes.

![INLET TEMPERATURE PROFILE - RUN 22](image)

**Figure A.7 The Inlet Temperature Profile for the Un-Cooled Radial Profile**

In Figure A.7 the inlet temperature profile for Run 22, which is the un-cooled radial profile, is shown as an average of the two upstream rakes. The red circles show the
actual measurements by the thermocouples, and the black circles show the two predicted values obtained at 5% and 0% span locations by extending this profile. The same layout will be followed for the upcoming figures as well. For the predictions, the closest additional rake measurement to these two locations that was obtained for another research program was used. This program utilized a very similar heat exchanger system, resulting in a very similar inlet temperature profile. This additional measurement was below 10% span, and was scaled to accommodate for the profile of Run 22. 5% and 0% predictions were obtained by applying a linear extrapolation onto this scaled value.

As mentioned before, the closest rake to the instrumentation and its corresponding span locations were used for Stanton Number calculations. However, the predictions for 0% and 5% spans are obtained using the average temperature profile shown in Figure A.7 from the two rakes. It is assumed that these predicted 0% and 5% span values are equal to the corresponding spans on one single rake that is closest to the endwall and 5% span gauges.

![Figure A.8 The Influence of Extended Inlet Temperature Profile on 5% Span Stanton Numbers](image-url)
Figure A.9 The Influence of Extended Inlet Temperature Profile on Inner Endwall Stanton Numbers

Figure A.8 shows the influence of using the 5% rake prediction instead of the 10% rake measurement on Stanton Numbers at 5% span of the airfoil. There is a gradual growing difference between the two cases towards the trailing edge. The predictions suggest higher Stanton Numbers at all wetted distance locations. This is because of the driving temperature getting smaller due to use of smaller upstream predicted
temperatures. The driving temperature further gets smaller towards the trailing edge due to the heating up of the flow with acceleration. This results in almost a 0.0005 difference in Stanton Numbers, while it is only about 0.0001 at the leading edge region. There is of course a growing uncertainty in Stanton Number calculations with decreasing driving temperature, which should be also kept in mind while interpreting these results. If the inlet temperatures were set to be much higher, this type of an analysis would give more accurate comparisons. However, it still provides a good approximation of by how much Stanton Numbers differ due to using local temperatures in calculations.

Figure A.9 shows the same type of comparison for the inner endwall. This time, the prediction at 0% rake is used to calculate Stanton Numbers. In data analysis, it was always 10% rake measurement that was used for both the endwall and 5% span locations on the airfoil. Again, there is generally an increase observed with using the predicted temperature at 0%. The difference between the two cases is generally around 0.0004 through the passage, while it is doubled at the endwall exit due to higher flow temperatures observed here. The differences between the two cases have increased at the endwall compared to 5% span due to the even smaller local upstream temperature used in calculations.

It is observed that there is up to a 0.0005 difference at 5% span, and up to a 0.0008 difference at the endwall in Stanton Numbers. This is a result of the temperature gradient of the radial temperature profile in this region. For a uniform temperature profile, the Stanton Numbers would not be affected as much with an extension in the temperature profile. This result is in parallel to the observation made in Section 1: the accuracy of the temperature measurement is critical in the flow regions with high temperature gradients.

The goal of the Appendix was to introduce possible Stanton Number definitions, and to present the differences between these definitions. The calculations of Section 1 and Section 3 point out to the fact that the temperature gradient causes significant changes in Stanton Numbers. Since there is a better collapse of the measurements with the use of the local temperature, and the local temperature measurements reflect the upstream profile effects as long as there is no disturbance in the flow field [116], the local
temperature definition should be used instead of the average rake temperature. On the other hand, the effects of using different cooling streams in the definition do not result in observable differences. These observations are helpful in making the decision for how to calculate Stanton Number given a variety of parameters, and are important since the data interpretation is based on Stanton Number trends observed throughout the vane section.