Secondary Flow, Turbulence, and Film Cooling Measurements in a Gas Turbine Vane Passage Downstream of a Novel Combustor-turbine Interface

A Dissertation

Submitted to the faculty of the University of Minnesota by
Kedar Prasad Nawathe

In partial fulfillment of the requirements for the degree of Doctor of Philosophy

Advised by Prof. Terrence W. Simon

April 2022
Acknowledgements

I would like to start by thanking my wonderful adviser, Prof. Terrence Simon. When I was considering continuing to a PhD after my master’s, I decided that I will do it only if it is under him as I knew that I will never find a better adviser anywhere else in the world. His intellect, coupled with his mighty experience, still surprises me and at the same time makes me wonder how someone like him can be so humble. He is not just an awesome adviser, but one of the best human beings I have ever met. I am indebted to him for every good thing that has happened to me in my grad school life these past five years.

I want to thank Solar Turbines Inc. for providing the financial support for my project and leaving me room to do some research to appease my own curiosity. Specifically, I want to thank Dr. Yong Kim whose insight in the gas turbine film cooling field has helped me immensely in conducting the wind tunnel experiments. I also want to thank the University of Minnesota Graduate School for providing financial support for my final year of grad school by awarding me the Doctoral Dissertation Fellowship. The fellowship gave me ample time to properly finish my thesis and to pursue some additional research.

My research group members have been an extremely important factor in finishing my degree. I will start by thanking Dr. Vlad Goldenberg, whose knowledge of turbomachinery and manner of presenting research left a mark on me. I want to thank Dr. Xinyi Li for her help while recording the measurements and providing valuable comments with a perspective of an outsider to the gas turbines field. I am grateful to Aaditya Nath for helping me in the lab in all aspects despite his busy schedule, providing both professional and personal help during the pandemic, and for giving me an insight into the theory of music. I want to thank Catherine Curtin for our discussions on her project as they made me think up some clever ideas in my own research. I would like to thank Vikas Jukanti and Sergio Martinez Pajuelo for our short yet pleasant time together. Finally, I want to thank Dr. Mahmood Alqefl, for his continued interest and feedback on my work. His inputs were invaluable in dissecting the complex data.
I would also like to thank my PhD preliminary and final exam committee, Prof. Sungyon Lee, Prof. Shri Ramaswamy, and Prof. Vinod Srinivasan, for their comments on my research plans. One of their comments in my oral preliminary exam led to a full-blown study (presented in Chapter 6), which I would say is one of the cooler parts of this thesis.

I had the opportunity to work with a lot of the professors in the department as a teaching assistant over the last few years. Each of them taught me something that has made my research work better. May it be the utmost dedication and tenacity toward their research, or may it be the technique of explaining a difficult concept easily to an unfamiliar audience. While being a TA, I was able to look at the fundamentals in a way that I never had before, and it has helped me many times in my research. So, in the order in which I was a TA for them, I would like to thank the professors that I have been a TA for: Prof. Al Aksan, Prof. Matt Anderson, Prof. Terrence Simon, Prof. Kim Stelson, Prof. Xiaojia Wang, Prof. Sungyon Lee, and Prof. Frank Kelso. It was an absolute delight to work with all of them.

When I was preparing for qualifying exams, I thought at times that I was not ready for them. Thankfully, I had my study group to rely on for support. These people helped me not go more insane than I already was during the scary Fall 2019 semester and hence, require a special mention. I want to thank everyone for what they helped me with the most: Zak Kujala for his perspective on fluids and the much-needed puns, Rana Bachnak and Zih-Yin Chen for bringing those special fluids problems to our meetings that ultimately made my fluids quals the easiest, Chi Zhang for challenging me on my definitions of efficiencies and having my back while we were TAs together for the heat transfer course and to Matt Chosa, for being the calm in the storm that was our group and for being the first person with whom I could discuss Malazan Book of the Fallen.

My friends have played a big role in helping me keep a good work-life balance. I would like to thank Daniel Mendonza and Vinamra Palaiya, my ex-roomies and close friends, who tolerate my overly logical approach in our conversations and with whom I always enjoy discussing good movies, TV shows, and prog music. I want to thank Nitish Ponkshe for reigniting my interest in cricket and for some most
interesting 1 AM conversations. Thank you, Manish Sachdeva, for helping me survive that especially demanding TA course, for advising me to plot the vorticity contours, and in general, for being there when I needed help. Thank you, Ashwin Karanam, for helping me in making the doctoral dissertation fellowship application and for encouraging me to finish Dark Souls III.

Even with having all these people around me, there were some parts of this journey (arguably, the toughest) that I had to tread alone. These mainly consisted of the grueling 10-hour night shifts for the experiments in a 100-dB room, the pandemic of 2020, and the writing of this thesis. These times were made bearable almost singlehandedly by a musician, Devin Townsend. His prolific music, podcasts on his creative process, his autobiography, and his outlook on life have been a huge help in the last few years. So, thank you Devin, for Terria, for Empath (even the B-side), and most importantly, for Ghost.

I would not even have thought of going down this road without my family’s constant and often, irrational faith that I will succeed. My parents have always been understanding, supportive, and have made sure that I knew that they were always with me. Thank you, Aai and Baba, for helping me rise to the challenge I had set myself. My sister, Mugdha, has always been my rock and my role model, who at one time, quite literally came running to Minneapolis to help me during a tough situation. I doubt these years could have been as pleasant without her. And, my brother-in-law, Aditya, has always helped me alongside Mugdha in every step of the way. For that, I want to sincerely thank both of them. Finally, I want to thank my favorite family member, my three-year old niece, Charvi. I yearned for our video calls and even at the bleakest of times, only a thought of her was enough to put a smile on my face.
Dedicated to Aai and Baba,

Thank you for the endless sacrifices.
# Table of Contents

List of Figures ................................................................................................................. x
List of Tables .................................................................................................................... xvii
Nomenclature ....................................................................................................................... xviii

## Chapter 1. Introduction and Literature Review ......................................................... 1

1.1. Vane Passage Aerodynamics .................................................................................. 3
   Secondary Flow Physics ............................................................................................... 3
   Secondary Flow Suppression ....................................................................................... 5

1.2. Endwall Film Cooling ............................................................................................ 8
1.3. Interactions of Multiple Coolant Flows ................................................................ 13
   Effects on Secondary Flow Physics and Endwall Cooling ......................................... 14
   Effects on Vane Surface Phantom Cooling ................................................................ 17

1.4. Decay of Turbulence through Vane Passage ....................................................... 19
1.5. Organization of the Thesis ..................................................................................... 21

## Chapter 2. Experimental Facility Description .......................................................... 23

2.1. Overview ............................................................................................................... 23
2.2. Mainstream Flow Supply ....................................................................................... 24
2.3. Turbulence Generator ........................................................................................... 25
2.4. Nozzle .................................................................................................................... 27
2.5. Test Section ........................................................................................................... 28
   2.5.1. Combustor-turbine Interface ............................................................................ 30
   2.5.2. Contoured Endwall ......................................................................................... 31
   2.5.3. Vane Cascade ................................................................................................. 33
   2.5.4. Test Section Access ....................................................................................... 35
2.6. Exit Flow Management ......................................................................................... 39
2.6.1. Tailboards ........................................................................................................ 39
2.6.2. Diffuser ............................................................................................................ 40

Chapter 3. Instrumentation ................................................................................. 42
3.1. Thermocouple .................................................................................................. 42
3.2. Inclined Manometer ...................................................................................... 43
3.3. Pressure Transducer ...................................................................................... 44
3.4. Five-hole Pressure Tube ................................................................................ 46

Calibration Procedure .......................................................................................... 47
Velocity Components Calculation Procedure ....................................................... 51

3.5. Laminar Flowmeter ....................................................................................... 52
3.6. Hot-wire Anemometry ................................................................................... 53

Overview of Calibration Procedure .................................................................... 55

Chapter 4. Test Section Qualification ................................................................. 58
4.1. Reynolds Number of the Vane Cascade Flow .............................................. 58
4.2. Flow Periodicity between Vane Passages .................................................... 59
4.3. Approach Flow Velocity and Turbulence Features ...................................... 60

4.3.1. Bulk Flow Features .................................................................................. 61
4.3.2. Turbulence Features ................................................................................. 64

Integral Length Scale, $\Lambda$ ............................................................................... 67
Energy Length Scale, $Lu$ ..................................................................................... 69
Taylor Microscale, $\lambda$ ...................................................................................... 70

4.4. Thermal Qualification Results ....................................................................... 72
4.4.1. Adiabaticity of Vane Passage Surfaces .................................................... 72
4.4.2. Approach Flow Thermal Profile ............................................................... 73
Chapter 5. Performance of the Close-coupled Combustor-turbine Interface Geometry........................................................................................................ 74

5.1. Motivation............................................................................................................. 74

5.2. Experimental Procedure..................................................................................... 74

  5.2.1. Experimental Cases....................................................................................... 74
  5.2.2. Aerodynamic Measurements Procedure...................................................... 75
  5.2.3. Thermal Measurements Procedure............................................................... 79
         Coolant Concentration Measurements............................................................ 80
         Surface Effectiveness Measurements.............................................................. 84

5.3. Vane Passage Flowfield Measurements ......................................................... 86

  Plane 1 (X/Cax = -0.104) ...................................................................................... 87
  Plane 2 (X/Cax = -0.02, Passage Inlet Plane)....................................................... 88
  Plane 3 (X/Cax = 0.289) ....................................................................................... 93
  Plane 4 (X/Cax = 0.612) ....................................................................................... 101
  Plane 6 (Throat Plane; starts at X/Cax = 0.612 at the Suction Surface)................................. 108
  Plane 5 (X/Cax = 1.02, Passage Exit Plane).............................................................. 114

5.4. Vane Passage Surface Effectiveness Measurements......................................... 119

         Endwall Adiabatic Cooling Effectiveness......................................................... 119
         Pressure Surface Adiabatic Cooling Effectiveness........................................... 125
         Suction Surface Adiabatic Cooling Effectiveness.............................................. 129

5.5. Conclusions........................................................................................................... 132

Chapter 6. Evolution of Secondary Flow Systems due to Changes in
Combustor Coolant Flowrates .................................................................................. 136

6.1. Motivation............................................................................................................. 136

6.2. Experimental Procedure..................................................................................... 136

6.3. Results and Discussion....................................................................................... 138

  6.3.1. Repeatability in Generation of Secondary Flows........................................... 138
6.3.2. Effect of Changing the Reference Pressure of Measurements.. 141
6.3.3. Discussion of Secondary Flowfield Measurements ................... 145

   Plane 3 (X/Cax = 0.289) ....................................................................... 146
   Plane 4 (X/Cax = 0.612) ....................................................................... 151
   Plane 5 (X/Cax = 1.02, Passage Exit Plane) ........................................ 156

6.4. Conclusions............................................................................................... 162

Chapter 7. Decay of Turbulence in a Vane Passage ......................... 164
7.1. Motivation.................................................................................................. 164
7.2. Theory of Turbulence ............................................................................. 164
   7.2.1. Characterization of Bulk Flow Properties .................................. 165
   7.2.2. Spectral Distribution of Turbulence Kinetic Energy .................... 167
   7.2.3. Length Scales of Turbulence ..................................................... 171
      Integral Length Scale, $\Lambda$ ............................................................ 171
      Taylor Microscale, $\lambda$ ................................................................. 175
      Kolmogorov Scale, $\eta$ ................................................................. 176
      Other Scales of Turbulence ............................................................ 177
7.3. Experimental Procedure........................................................................ 179
7.4. Numerical Simulations Procedure ....................................................... 182
   7.4.1. Turbulence Closure Models ....................................................... 182
   7.4.2. Simulations Setup ...................................................................... 184
7.5. Results and Discussion ........................................................................ 187
   7.5.1. Bulk Flow Properties .................................................................. 191
   7.5.2. Power Spectral Distribution ...................................................... 195
      Axial Variation in Spectral Distribution .......................................... 195
      Pitchwise Variation in Spectral Distribution .................................. 196
   7.5.3. Rate of Turbulence Dissipation ............................................... 198
   7.5.4. Length Scales of Turbulence ..................................................... 201
Integral Length Scale ................................................................. 201
Remaining Turbulence Scales .......................................................... 207
Axial Variation in Turbulence Length Scales............................... 208
7.6. Conclusions ................................................................................. 209

Chapter 8. Concluding Remarks ......................................................... 211
8.1. Summary ...................................................................................... 211
8.2. Future Work .................................................................................. 213

References ............................................................................................. 215

Appendix A. Uncertainty Analysis ......................................................... 223
A.1. Endwall Effectiveness and Temperature Recovery Coefficient ....... 224
A.2. Velocities Measured by Five-hole Probe ...................................... 225
      Uncertainty in Absolute Velocity ................................................. 226
      Uncertainty in Flow Vectors Projection ....................................... 226
A.3. Velocities Measured by Hot-wire Anemometer ......................... 227
List of Figures

Figure 1.1.1 Secondary flow system in vane passages [1]......................... 3
Figure 1.1.2 Formation of recirculation zones upstream of an obstacle [2] .... 4
Figure 1.1.3 Formation of horseshoe vortex around a cylindrical obstacle [3]...... 5
Figure 1.1.4 Most comprehensive secondary flow field in the vane passage by Wang et al. [5] .......................................................... 6
Figure 1.1.5 Various endwall contours compiled by Burd [7]. Originally extracted from [8,9,10,11,12,13,14] .................................................. 7
Figure 1.2.1 Employing multiple cooling rows throughout the vane passage. Top: Thermocouple and cooling rows location. Bottom: Cooling effectiveness contours [18] .......................................................... 8
Figure 1.2.2 Cooling hole configuration studied by Jabbari et al. (a) Location and orientation of cooling holes, (b) Effectiveness results of blowing ratios (M) of 1.0 (left) and 2.0 (right) [19] ........................................................................ 9
Figure 1.2.3 Parametric study on endwall cooling effectiveness performed by Thrift et al. (a) Cases with 90° slot orientation, (b) Cases with 45° slot orientation [20] 10
Figure 1.3.1 Counter-rotating vortex in the study by Colban et al. [32]............ 13
Figure 1.3.2 Counter-rotating vortex in the study by Alqefl et al. [37] .............. 14
Figure 1.3.3 Phantom cooling of suction surface for different velocity ratios [45] 17
Figure 1.3.4 Phantom cooling in presence of combustor coolant (left) pressure surface (right) suction surface [41] .................................................. 18
Figure 2.1.1 Test facility overview [67] ...................................................... 23
Figure 2.3.1 Turbulence generator (Original drawing from [67]) .................... 25
Figure 2.3.2 Turbulence generator design specifications [67] ....................... 26
Figure 2.4.1 Wind tunnel nozzle [67] ......................................................... 27
Figure 2.4.2 Nozzle contours [64] .............................................................. 27
Figure 2.5.1 Top view of the experimental test section Error! Bookmark not defined.
Figure 2.5.2 Front view of the experimental test section ................................. 29
Figure 2.5.3 Contoured endwall in reference to the vanes [67] ........................ 32
Figure 2.5.4  Vane assembly with the vane slabs (Left) Model, (Right) During assembly [66] ........................................................................................................................................ 35
Figure 2.5.5 Jig for drilling holes on vane surfaces (Left) In reference to the vane surface (Right), dimensional specifications [66] ........................................................................... 35
Figure 2.5.6 Static pressure tap locations along the vane surfaces [66] .............. 37
Figure 2.5.7 Test section access region. Four rigid panels and the movable hole panel are indicated. [67] .................................................................................................................. 38
Figure 2.5.8 Movable-hole panel [63] ........................................................................ 39
Figure 2.6.1 Different views of the diffuser [67] ...................................................... 40
Figure 3.2.1 Inclined manometer used in this study [67] ........................................... 44
Figure 3.4.1 Schematic of the conical five-hole probe used in this study (provided by United Sensors Corp.) ........................................................................................................ 47
Figure 3.4.2 Test section axes. The main flow moves in the positive axial direction. ................................................................................................................................. 48
Figure 3.4.3 Five-hole probe calibration setup. Original drawing referenced from [68]. ................................................................................................................................. 49
Figure 3.6.1 Dimensions of the model 1218 standard boundary layer hot-wire probe used in this study [74] ........................................................................................................... 54
Figure 3.6.2 Hot-wire calibration setup [66] ................................................................ 55
Figure 3.6.3 Calibration curve of the hot-wire probe used in this study .......... 56
Figure 4.2.1 Static pressure profile of vane surfaces ............................................... 60
Figure 4.3.1 Axial location of the approach flow characterization plane .............. 61
Figure 4.3.2 Velocity measurement locations for the approach flow characterization plane ..................................................................................................................... 62
Figure 4.3.3 Mean velocity contours at the approach flow characterization plane ............................................................................................................................. 63
Figure 4.3.4 RMS velocity fluctuations at the approach flow characterization plane ............................................................................................................................. 64
Figure 4.3.5 Turbulence intensity distribution at the approach flow characterization plane.......................................................................................................................... 65
Figure 4.3.6 Power spectral density spectra measured at mid-pitch midspan location the approach flow characterization plane .............................................. 68
Figure 5.2.1 Isometric view of endwall-vane connecting region ....................... 76
Figure 5.2.2 Axial locations of aerodynamic measurement planes. X/Cax values are indicated for each plane near the suction surface .............................................. 77
Figure 5.2.3 Aerodynamic measurement locations ........................................... 79
Figure 5.2.4 Axial locations of thermal measurement planes. X/Cax values are indicated for each plane near the suction surface .............................................. 81
Figure 5.2.5 Thermal measurement locations ................................................... 82
Figure 5.2.6 Surface measurement locations ................................................... 85
Figure 5.3.1 Plane 1: Thermal contours ......................................................... 87
Figure 5.3.2 Plane 2: Absolute velocity contours .............................................. 89
Figure 5.3.3 Plane 2: Vorticity contours with secondary flow vectors .................. 90
Figure 5.3.4 Plane 2: Thermal contours ......................................................... 91
Figure 5.3.5 Plane 2: Thermal contours with secondary flow streamlines ......... 92
Figure 5.3.6 Plane 3: Absolute velocity contours ............................................ 95
Figure 5.3.7 Plane 3: Vorticity contours with secondary flow vectors ............... 96
Figure 5.3.8 Plane 3: Thermal contours ......................................................... 97
Figure 5.3.9 Plane 3: Thermal contours with secondary flow streamlines ......... 98
Figure 5.3.10 Plane 4: Absolute velocity contours ......................................... 102
Figure 5.3.11 Plane 4: Vorticity contours with secondary flow vectors ............. 104
Figure 5.3.12 Plane 4: Thermal contours ....................................................... 105
Figure 5.3.13 Plane 4: Thermal contours with secondary flow streamlines ...... 107
Figure 5.3.14 Plane 5: Absolute velocity contours ......................................... 110
Figure 5.3.15 Plane 5: Vorticity contours with secondary flow vectors ............. 111
Figure 5.3.16 Plane 5: Thermal contours ....................................................... 112
Figure 5.3.17 Plane 5: Thermal contours with secondary flow streamlines ...... 113
Figure 5.3.18 Plane 6: Absolute velocity contours ......................................... 115
Figure 5.3.19 Plane 6: Vorticity contours with secondary flow vectors ............. 116
Figure 5.3.20 Plane 6: Thermal contours ....................................................... 117
Figure 5.3.21 Plane 6: Thermal contours with secondary flow streamlines ...... 118
Figure 5.4.1 Endwall adiabatic cooling effectiveness of current study......... 121
Figure 5.4.2 Endwall adiabatic cooling effectiveness of previous study [40]. The
case names in parenthesis are the equivalent cases from the current study. .. 122
Figure 5.4.3 Pressure surface adiabatic cooling effectiveness of current study 126
Figure 5.4.4 Pressure Surface adiabatic cooling effectiveness of previous study
[40]. The case names in parenthesis are the equivalent cases from the current
study. ................................................................................................................ 127
Figure 5.4.5 Suction surface adiabatic cooling effectiveness of current study.. 130
Figure 5.4.6 Suction Surface adiabatic cooling effectiveness of previous study [41].
The case names in parenthesis are the equivalent cases from the current study.
......................................................................................................................... 131
Figure 5.5.1 Overall vane passage adiabatic cooling effectiveness. Note that the
vanes extend in the positive Z-direction up to 50% of the inlet span. ............ 135
Figure 6.2.1 Axial locations of measurement planes. X/Cax values are indicated
for each plane near suction surface................................................................. 137
Figure 6.2.2 Aerodynamic measurement locations........................................ 138
Figure 6.3.1 Plane 4 Case C26 vorticity contours. These were recorded to verify
the repeatability of generated secondary flows ............................................. 140
Figure 6.3.2 Sudden shift in secondary flow vectors on Plane 5 ..................... 141
Figure 6.3.3 Rerecording a fraction of measurements of Case C26 for verifying
effect of changing reference pressure. Two separate probe insertion angles were
considered ....................................................................................................... 143
Figure 6.3.4 Replotting of Plane 5 measurements for Cases C26 and C35 by using
two different ideal streamline vectors ........................................................... 144
Figure 6.3.5 Plane 3 velocity contours .............................................................. 146
Figure 6.3.6 Plane 3 vorticity contours with secondary flow vectors and
streamlines: Cases C00 and C26................................................................. 147
Figure 6.3.7 Plane 3 vorticity contours with secondary flow vectors and
streamlines: Cases C35 and C46................................................................. 148
Figure 6.3.8 Plane 3 vorticity contours with secondary flow vectors and
streamlines: Cases C56 .................................................................................. 149
Figure 7.5.9 Pitchwise variation in spectral distribution for plane 3 at Z/S\textsubscript{local} = 0.5 ................................................................. 198

Figure 7.5.10 Pitchwise variation in spectral distribution for plane 3 at Z/S\textsubscript{local} = 0.664 ................................................................................................................ 198

Figure 7.5.12 Pitchwise Variation in re-normalized spectral distribution for plane 3 at Z/S\textsubscript{local} = 0.664 ................................................................................................................ 199

Figure 7.5.11 Pitchwise Variation in re-normalized spectral distribution for plane 3 at Z/S\textsubscript{local} = 0.5 .................................................................................................. 199

Figure 7.5.13 Experimental and numerical turbulence dissipation [82]........... 200

Figure 7.5.14 Plane 1: Pitchwise variation in parameters influencing integral scales (a) TKE at Lowest Frequency, (b) RMS Velocity Fluctuations, (c) Integral Length Scale, (d) Integral Time Scale ................................................................. 202

Figure 7.5.15 Plane 2: Pitchwise variation in parameters influencing integral scales (a) TKE at Lowest Frequency, (b) RMS Velocity Fluctuations, (c) Integral Length Scale, (d) Integral Time Scale ........................................................................ 203

Figure 7.5.16 Plane 3: Pitchwise variation in parameters influencing integral scales (a) TKE at Lowest Frequency, (b) RMS Velocity Fluctuations, (c) Integral Length Scale, (d) Integral Time Scale ........................................................................ 204

Figure 7.5.17 Plane 4: Pitchwise variation in parameters influencing integral scales (a) TKE at Lowest Frequency, (b) RMS Velocity Fluctuations, (c) Integral Length Scale, (d) Integral Time Scale Note that there are only three pitch locations measured on this plane. ................................................................. 205

Figure 7.5.18 Plane 5: Pitchwise variation in parameters influencing integral scales (a) TKE at Lowest Frequency, (b) RMS Velocity Fluctuations, (c) Integral Length Scale, (d) Integral Time Scale ........................................................................ 206

Figure 7.5.19 Experimental integral length scales ........................................ 207

Figure 7.5.20 Comparison of measured and computed Taylor microscales..... 207

Figure 7.5.21 Comparison of measured and computed Kolmogorov scales .... 208

Figure 7.5.22 Comparison of measured and computed energy length scales.. 208
Figure A.2.1  A typical projection of velocity vector seen in plotted results. The length of the vector is equal to the magnitude of the velocity and $\beta$ is the angle the vector makes with z-axis. ................................................................. 225
List of Tables

Table 2.5.1 Cascade vane geometric specifications ................................................. 34
Table 3.4.1 Details of the contour fits ..................................................................... 51
Table 4.3.1 Approach flow characteristics .............................................................. 72
Table 4.3.2 Comparison of turbulence characteristics .............................................. 72
Table 5.2.1 Experimental cases ........................................................................... 75
Table 5.2.2 Difference in the static and stagnation temperatures at different velocities .......................................................... 83
Table 5.2.3 Surface measurement locations .......................................................... 86
Table 6.2.1 Test matrix ....................................................................................... 137
Table 7.4.1 Boundary conditions for numerical simulations ..................................... 187
Table 7.5.1 Dissipation rates at all planes ............................................................... 199
Table A.1.1 Uncertain parameters in the temperature measurements ................... 224
Table A.2.1 Uncertain parameters in the in-field velocity measurements ............. 226
Table A.3.1 Uncertain parameters in the high-frequency approach flow velocity .......................................................... 227
Table A.3.2 The uncertainties of various measured parameters ............................ 228
## Nomenclature

### Symbols

- $A, B, n$: Hotwire calibration constants
- $C$: Vane true chord length [cm]
- $C_{ax}$: Vane axial chord length [cm]
- $C_p$: Pressure coefficient [-]
- $E(\omega)$: Energy density spectrum [m$^2$.s$^{-2}$.Hz$^{-1}$]
- $L_u$: Energy length scale of turbulence [cm]
- $u'$: Root mean square of velocity fluctuations [m.s$^{-1}$]
- $g$: Acceleration due to gravity [m.s$^{-2}$]
- $H$: Inlet passage height [cm]
- $H_c$: Height of the combustor exit [cm]
- $k$: Turbulence kinetic energy [m.s$^{-2}$]
- $MFR$: Ratio of coolant mass flow rate to mainstream mass flow rate [-]
- $N$: Sample size [-]
- $P$: Cascade pitch (spacing between successive vanes) [cm] or Pressure [Pa]
- $Q$: Volumetric flow rate [m$^3$.s$^{-1}$]
- $Re$: Reynolds number [-]
- $S$: Vane or approach plane span [cm]
- $T$: Absolute temperature [K]
- $Tu$: Turbulence intensity [%]
- $u(t)$: Instantaneous velocity [m.s$^{-1}$]
- $u'(t)$: Instantaneous fluctuation about time-averaged mean velocity [m.s$^{-1}$]
- $\bar{U}$: Time-averaged mean velocity in axial direction [m.s$^{-1}$]
- $U_0$: Approach flow velocity [m.s$^{-1}$]
- $V$: Velocity component [m.s$^{-1}$] or Voltage [V]
- $f$: Frequency of energy density spectrum [Hz]
\( X \) or \( x \)  
Passage axial direction

\( Y \)  
Passage pitchwise direction

\( Z \)  
Passage spanwise direction

**Greek**

\( \rho \)  
Density [kg.m\(^{-3}\)] or Autocorrelation function [-]

\( \Delta \)  
Difference operator

\( \varepsilon \)  
Turbulence kinetic energy dissipation rate [m\(^2\).s\(^{-3}\)]

\( \eta \)  
Surface adiabatic cooling effectiveness [-] or Kolmogorov length scale [mm]

\( \Lambda \)  
Turbulence integral length scale [cm]

\( \lambda \)  
Taylor microscale [cm]

\( \alpha \)  
Pitch angle [°]

\( \theta \)  
Recovery temperature coefficient [-]

\( \mu \)  
Dynamic viscosity [N.s.m\(^{-2}\)]

\( \nu \)  
Kinematic viscosity [m\(^2\).s\(^{-1}\)]

\( \Omega \)  
Vorticity [s\(^{-1}\)]

\( \gamma \)  
Yaw angle [°]

\( \tau \)  
Time interval between two measurements [s]

\( \omega \)  
Specific dissipation rate (s\(^{-1}\))

**Subscripts**

\( ave \)  
Average

\( e \)  
Located at passage exit plane

\( C \)  
Combustor coolant or coolant

\( chord \)  
Based on vane true chord length

\( exit \)  
Passage exit

\( F \)  
Film coolant flow property

\( i \)  
Located at passage inlet plane

\( local \)  
At the location of the measurement under consideration
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$s$</td>
<td>Relative to streamwise direction or sample</td>
</tr>
<tr>
<td>$stag$</td>
<td>A stagnation property</td>
</tr>
<tr>
<td>$static$</td>
<td>A static property</td>
</tr>
<tr>
<td>$T$</td>
<td>With respect to time</td>
</tr>
<tr>
<td>$X$ or $u$</td>
<td>Relative to passage axial direction</td>
</tr>
<tr>
<td>$Y$</td>
<td>Relative to passage pitchwise direction</td>
</tr>
<tr>
<td>$Z$</td>
<td>Relative to passage spanwise direction</td>
</tr>
<tr>
<td>$\infty$</td>
<td>Mainstream flow property</td>
</tr>
</tbody>
</table>
Gas turbines have been around for nearly a century. Their importance in the aviation and energy generation sectors has increased rapidly since their inception. A significant amount of research is being done to increase the efficiency of the engine and to decrease the negative effects it has on the environment. One way to increase the efficiency is to increase the combustor exit temperature. This exit temperature is usually much higher than the melting temperature of the turbine components located downstream of the combustor. Therefore, sophisticated cooling schemes are needed. As the coolant flow is drawn from the compressor section of the engine, increasing the cooling efficiency by minimizing the amount of coolant usage becomes an important topic of research. The cooling is most important in the first stage of the vanes that are immediately downstream of the combustor. The endwall and the vane surfaces must be cooled aggressively to avoid thermal failure. One of the techniques used to cool these surfaces is to introduce cooler air through rows of holes that will create a coolant film. Usually, a particular set of cooling holes is designed to specifically cool only one surface. Most of the studies in the literature focus on only one such set of cooling holes. However, as the coolant flows have high momentum, they are bound to interact with each other and change the fluid dynamics inside a vane passage. These aerodynamic interactions between different cooling flows largely remain un researched. As the coolant transport establishes how well the coolant will cool a particular vane passage surface, it is essential to understand the complex physics behind the interaction of these coolant flows.

The first study in this thesis reports experiments conducted in a test facility that employed a new design of the combustor-turbine interface. It was designed in a way to provide optimum cooling to both the combustor wall and the endwall. The experiments were conducted for a variety of combustor coolant mass flow rates. Aerodynamic flowfield and coolant concentration measurements were recorded to understand how the flow physics is affected by the new coolant injection scheme for different coolant flow rates. Thermal measurements were taken along the
endwall and vane surfaces throughout the vane passage to document the adiabatic effectiveness. The main objective of this study is to show that an optimum utilization of the combustor coolant can decrease the overall coolant requirement leading to an increase in the efficiency of the gas turbine engine. The results of this study will be compared with previous studies taken on the same facility that had an engine-representative combustor-turbine interface. This comparison is expected to highlight the advantages of the new interface employed in the current studies. After these measurements were finished, it was found that this new coolant scheme can be used to study the transition of one type of secondary flow system to another type of secondary flow system. Therefore, a second study was designed to document and explain this transition. These two studies, together, provide a detailed performance analysis of the new coolant injection scheme to help gas turbine designers to improve the cooling performance of vane passage surfaces.

A third study was conducted that involved measuring the decay of turbulence through the vane passage and its effects on the turbulence features. These measurements were taken in the same facility as the earlier studies. Suitable experimental data showing this decay are lacking in the literature. The turbulence closure models used to numerically analyze gas turbine flows are known to not predict the turbulence decay satisfactorily. Hence, the experimental measurements are compared with simulations done on the same geometry using different turbulence models to understand where and why the models fail, which would help the turbulence modelers in improving their existing models for gas turbine applications.

Fluid flow in the vane passage has always been complex and is further affected by various coolant injection schemes designed in the past few decades. To explain the methods behind the design of the current test facility and to show the state of the art, the background on vane aerodynamics and film cooling techniques is given. The various topics involving this subject will now be addressed.
1.1. Vane Passage Aerodynamics

Secondary Flow Physics

The vane geometry creates a pressure gradient in the passage between successive vanes. The change of cross-sectional area in the mainstream flow direction leads to a rapid acceleration followed by a mild deceleration in the vane passage. These conditions develop complex flow structures in the passage. These flow structures consist of fluid moving in directions other than the ideal streamline direction and they are collectively called secondary flows. As secondary flows are deviations from the ideal streamline, they cause pressure losses in the turbine stage leading to a decrease in efficiency. Langston et al. (1990) [1] presented one of the first studies on secondary flows. Figure 1.1.1 shows his interpretation of vane passage flow. He found three major secondary flow features: a horseshoe vortex, a passage vortex, and endwall crossflow. Their descriptions are given below.

At the stagnation line on the leading edge of the vane, the stagnation pressure in the mainstream is higher than the stagnation pressure in the boundary layer.

![Secondary flow system in vane passages](image)
along the endwall. This difference in stagnation pressures leads to a recirculation zone upstream of the leading edge. This recirculating flow travels downstream as two separate vortices. If looked at from the top (view of fig. 1.1.1), they resemble a horseshoe and thus, are called the horseshoe vortices. A horseshoe vortex can occur at any obstacle as reported by Sedney and Kitchens (1975) [2] with the obstacle being a wall. The formation of a recirculation zone for their geometry is shown in fig. 1.1.2. Goldstein and Karni (1984) [3] similarly studied a flow with a cylindrical obstacle. The recirculation zones observed in this study traveled downstream around the cylinder, resulting in a horseshoe shape to the flow when viewed from the top. This flowfield is shown in fig. 1.1.3.

Langston was able to see this horseshoe vortex at the vane leading edge. He found that the two legs of the horseshoe vortex behaved differently after entering the vane passage. The vortex along the suction surface of the vane, called the suction leg, remained along the suction surface as it moved through the passage. The pressure leg, however, experienced the pressure gradient present between the pressure and the suction surfaces of successive vanes and migrated toward the suction surface of the vane, as seen in fig. 1.1.1. This evolution of the pressure leg of the horseshoe vortex in the passage is called the passage vortex. This phenomenon was also confirmed by Sharma and Butler (1987) [4].
vortex, after reaching the suction surface, traversed alongside the suction leg of the horseshoe vortex. Additionally, due to the pressure gradient between the vane surfaces, a secondary flow from the pressure surface to the suction surface was seen along the endwall. This was termed as the endwall crossflow. Due to the direction of the endwall crossflow, it ends up helping the passage vortex, making it a major source of pressure loss. Wang et al. (1997) [5] performed flow visualization studies that gave a comprehensive description of the vane passage flow (fig. 1.1.4) that is generally accepted by most researchers.

Secondary Flow Suppression

A detailed description of vane passage pressure losses was given by Denton (1993) [6]. Considerable research has been done to find ways to suppress secondary flows, particularly losses due to the passage vortex. One of the important techniques is to replace the flat endwall with a contoured endwall. The secondary flow losses are highly dependent on the cascade geometry and a single contouring method cannot be established. Burd (1998) [7] provides a good compilation of different endwall contours as shown in fig. 1.1.5. Morris and Hoare (1975) [8] studied flow inside a vane passage with multiple endwall contours (shown in fig. 1.1.5 (A), (C), and (D)). They found that generally, endwalls with symmetric-cubic profiling located downstream of the most curved part of the vanes.

![Figure 1.1.3 Formation of horseshoe vortex around a cylindrical obstacle [3]](image-url)
(fig. 1.1.5 [A]) can reduce the secondary flow losses up to 25% for vanes with low aspect ratios and small inlet boundary layer thicknesses compared to the pressure losses in the vanes with flat endwalls (fig. 1.1.5 [B]). Similarly, Kopper et al. (1981) [9] performed experiments in a cascade that had a contoured endwall on one side and a flat endwall on the other. They were able to see up to 17% reduction in the aerodynamic losses due to the contouring.

Burd and Simon (2000) [15] recorded detailed aerodynamic measurements in a vane cascade with one endwall being flat and the other being contoured. They found that the passage vortex strength near the contoured endwall was lower than that of the flat endwall. A much smoother contouring was employed to the endwall in the experiments conducted by Piggush and Simon (2012) [16]. They also

Figure 1.1.4 Most comprehensive secondary flow field in the vane passage by Wang et al. [5]
reported the passage vortex being larger near the flat endwall compared to the contoured endwall. Most of the experimental studies involving contoured endwalls used axisymmetric contouring and found that contouring can significantly suppress secondary flows.

Figure 1.1.5 Various endwall contours compiled by Burd [7]. Originally extracted from [8-14].
1.2. Endwall Film Cooling

The vane passage surfaces are cooled by injecting cool air through an array of holes. This injection forms a coolant film on these surfaces. This technique is called film cooling and is extensively used to cool the endwalls and the vane surfaces. This section explains research studies involving adiabatic cooling of endwalls. Adiabatic cooling performance means that the surface to be cooled is considered to not have heat transfer through it. These studies try to explain how the injected coolant loses its ability due to mixing with the mainstream. The effect of the assumption of surface adiabaticity is discussed in section 5.2.3.

Figure 1.2.1 Employing multiple cooling rows throughout the vane passage. Top: Thermocouple and cooling rows location. Bottom: Cooling effectiveness contours [18]
Blair (1974) [17] was one of the first researchers to use this technique by injecting coolant through holes located upstream of the vane passage for cooling the endwall. He found that the passage vortex skewed the coolant toward the suction surface resulting in non-uniform cooling of the endwall. Takeishi et al. (1990) [18] added coolant injection rows in the vane passage to counter the coolant migration toward the suction surface (see fig. 1.2.1). Although it helped cool the endwall in the downstream part of the passage, the non-uniform coolant distribution was still observed. Based on the flow physics observed near the endwall, Jabbari et al. (1996) [19] designed a coolant hole configuration that is

![Cooling hole configuration](image)
shown in fig. 1.2.2 (a). They found that, for lower values of coolant-to-mainstream mass flux ratio, the mid-passage region of the endwall received good cooling while cases with higher mass flux ratios were helpful in cooling the downstream part of the endwall. For this coolant hole configuration, the passage vortex was observed in the mid-passage region, but its effects were diminished near the trailing edge region. Their results are shown in fig. 1.2.2 (b).

Figure 1.2.3 Parametric study on endwall cooling effectiveness performed by Thrift et al. (a) Cases with 90° slot orientation, (b) Cases with 45° slot orientation [20]
More recently, Thrift et al. (2012) [20] studied endwall cooling with coolant injection through a slot upstream of the vane passage for a variety of angles between the slot and the mainstream and for several coolant mass flow rate ratios (MFR). The mass flow rate ratio is defined as the ratio of mass flow rate of the coolant to mass flow rate of the mainstream. They found that for low injection angles and for values of MFRs below 1%, the endwall cooling performance increased with increasing coolant MFR. Figure 1.2.3 shows their results. Generally, the above studies have indicated that the location of coolant injection, the angle of injection, and coolant flow rates are important parameters to define a good endwall cooling scheme.

The above studies were conducted in the presence of a flat endwall. Burd and Simon (2000) [21,22] were among the first to study the effects of coolant injection on cooling effectiveness of a contoured endwall. They found that for low injection MFR values, secondary flows helped in concentrating the coolant near the suction surface side of the endwall. With increases in the MFR, the coolant momentum resisted the secondary flows and a significant amount of coolant remained near the pressure surface. Erickson and Simon (2011) [23] studied the performance of two endwall contours: dolphin nose shape and shark nose shape. They found that the endwall with a dolphin nose shaped contour always showed better cooling effectiveness than the shark nose shaped endwall, irrespective of the coolant flowrate. More recently, Ornano and Povey (2017) [24] studied the effect of coolant injection upstream of the vane passage in the presence of the effusion coolant flows that are introduced to cool the combustor liner walls. They discovered that an increase in the coolant momentum increased endwall cooling effectiveness up to a certain point. Any increase in coolant momentum beyond this value brought minimal increase in the cooling performance. They hypothesized that the coolant momentum increase helped in suppressing the passage vortex and led to performance increase. However, they did not record aerodynamic measurements that supported this claim.

Since coolant injection in the vane passage can change the flow physics, understanding its effect on secondary flows is important. Some of the earliest
studies in this field were done by Sieverding and Wilputte (1980) [25] and Goldman and McLallin (1977) [26]. They found that for a particular range of coolant MFRs, passage secondary flows can be significantly suppressed. They also found that low angles of injection were optimum for reducing secondary flows. Biesinger and Gregory-Smith (1993) [27] also found similar results. Additionally, they showed that for low blowing ratios, the coolant flows increased the horseshoe vortex intensity while for high blowing ratios, the coolant flows created their own secondary flows. Friedrichs et al. (1997) [28] reached similar conclusions. They showed that the spanwise distance up to which the secondary flows are affected by the coolant injection depend on the coolant MFR. They also found that the pressure losses due to individual rows of coolant injection could not be added proportionately as coolant flows injected from different rows interacted with each other.

The above studies were conducted on a flat endwall. Burd and Simon (2000) [21] studied the aerodynamic and thermal effects of coolant injection upstream of a vane passage with a contoured endwall. Similar to other studies, they found that moderate coolant blowing ratios suppressed secondary flows, with the suppression effects stronger near the contoured endwall than near the flat endwall. Piggush and Simon (2012) [16] studied the effects of coolant flows injected both upstream and in the vane passage (also called slash-face blowing) with a contoured endwall. They found that the vane passage pressure losses increase mainly due to the slash-face blowing. Schuepbach et al. (2010) [29] showed that coolant injection generally increased pressure losses, but the losses were more pronounced for a flat endwall than a contoured endwall, implying that a carefully profiled endwall contour is a good measure to reduce pressure losses. El-Gabry et al. (2015) [30] performed an experimental study that had coolant injection through upstream slots, slash-face slots, in-passage holes on the endwall, and holes on the vane surfaces. The coolant injected to cool the endwall had the most influence on the passage flowfield while the coolant flows injected to cool the vanes did not affect the passage flowfield considerably. Alqefl et al. (2017) [31] computationally showed that although upstream endwall film coolant injection
played a role in passage flow, the passage pressure losses due to the injection were not significant.

In general, the above studies suggest that a contoured endwall along with coolant injection upstream of the vane passage may either decrease or may not affect secondary flows in the passage. The results of these studies were used while designing the coolant injection schemes of the proposed study.

1.3. Interactions of Multiple Coolant Flows

Previous subsections have discussed the research done in cooling the endwall. However, only the endwall coolant flow was simulated in any of those studies. In the engine, all coolant flows introduced in the vicinity of the passage are expected to interact with each other. Their interactions are likely to influence the cooling of the passage surfaces. Hence, studies that consider more than one coolant source

![Figure 1.3.1 Counter-rotating vortex in the study by Colban et al. [32]](image)

Figure 1.3.1 Counter-rotating vortex in the study by Colban et al. [32]
have been conducted in the recent past to better predict the vane passage surfaces cooling performance.

Effects on Secondary Flow Physics and Endwall Cooling

Colban et al. (2003) [32,33] studied the effects of coolant flows introduced in the combustor section immediately upstream of the vane passage on coolant transport and endwall cooling. They found that the flow physics inside the vane passage was significantly modified due to consideration of combustor coolant. Specifically, near the vane pressure surface, a vortex that had a direction of rotation opposite to that of the passage vortex was seen in the velocity fields (fig.
1.3.1). However, an in-depth explanation behind the formation of this vortex was not provided. Such a vortex was also observed in a computational study of a similar vane cascade by Hermanson and Thole (2002) [34]. Saxena et al. (2016) [35] and Alqefl et al. (2018) [36] simulated thermal profiles upstream of the passage that matched the engine combustor exit dimensionless thermal profiles. In addition, they had coolant holes for endwall film coolant injection and a vane passage with a contoured endwall. They found that combustor coolant contributed significantly toward cooling the endwall and that increases in the MFR of endwall film coolant only marginally improved the cooling performance. However, aerodynamic aspects of combustor coolant flows were not simulated in the test facility and hence, the vortex seen by Colban et al. (2003) [32] was not observed.

Similar to the study mentioned in the preceding paragraph, a more comprehensive set of studies was conducted by Alqefl et al. (2021) [37-39]. In addition to the contoured endwall and endwall coolant injection, combustor coolant injection was also simulated. The facility generated engine-level Reynolds number and turbulence characteristics. The combustor coolant MFR was fixed while the endwall coolant MFR was varied for different cases. The study provided detailed aerodynamic and thermal descriptions of coolant transport and provided adiabatic endwall cooling effectiveness for all cases. The vortex reported by Colban et al. (2003) [32] was also observed in every case of this study (see fig. 1.3.2 for a representative flowfield at the vane passage exit). It was termed as an ‘impingement vortex’ and was found to be a result of the interactions of the combustor coolant with the endwall film coolant and the mainstream. The detailed mechanisms behind the formation of the impingement vortex are now discussed as this vortex plays a major role in the studies conducted as part of this thesis.

(a) **A difference in approach flow angles**: The combustor coolant flow has a high momentum and is injected axially (in the direction of the streamlines upstream of the vane passage). As the flow enters the passage, while the mainstream starts to change direction due to the passage turning owing to the vane geometry, the coolant flow attempts to stay axial. This generates a difference in the angles of
flow near the endwall and flow away from the endwall. This difference leads to a swirl that begins the formation of the impingement vortex at the passage inlet.

(b) Modified passage inlet velocity profile: In a vane cascade without combustor coolant injection, the boundary layer along the endwall at the leading edge is a generic turbulent boundary layer. Therefore, as the flow stagnates along the leading edge, the stagnation pressure in the flow near the endwall is lower than that of the mainstream which results in a secondary flow that travels from the mid-span toward the endwall (and forms a horseshoe vortex). In cascades that consider combustor coolant injection, the momentum of combustor coolant results in a higher stagnation pressure near the endwall than in the mainstream flow and this leads to a secondary flow from the endwall boundary layer toward the mid-span. This is the start of the formation of the impingement vortex. Similar to the horseshoe vortex, this vortex travels along the two vane surfaces, with the pressure side leg being significantly more dominant than the suction side leg.

(c) Continuous coolant impingement on pressure surface: The first two mechanisms start the creation of the impingement vortex at the leading edge. Downstream of the leading edge, the coolant momentum continues to resist the passage turning and moves axially. This causes the coolant to impinge on the pressure surface for most of the passage and energizes the impingement vortex. This helps to keep the impingement vortex near the pressure surface and to sustain itself throughout the vane passage.

Nawathe et al. (2021) [40,41] further worked on the same experimental test facility as Alqefl et al. (2021) [37-39] investigating the effects of changing the MFR of the combustor coolant. They reported that the flow physics, at large, did not change significantly with changing MFR, but the coolant transport near the endwall and vane pressure surface changed considerably as combustor coolant flow rate was changed. They also reported adiabatic cooling effectiveness along the endwall. Additionally, the combustor coolant was found to be providing significant cooling along the pressure surface, unintentionally. In general, it was found that moderate MFR values of both combustor coolant and endwall film coolant provide sustained cooling along the endwall as well as the vane pressure surface.
Effects on Vane Surface Phantom Cooling

The unintentional cooling of any vane passage surface is called second-order cooling or phantom cooling. In the context of this thesis, coolant that is introduced to cool the endwall can inadvertently end up cooling the vane pressure and suction surfaces. Hence, a review of the current literature on phantom cooling is given in this subsection.

Roback and Dring (1993) [42,43] were among the first to discuss the accumulation of coolant on vane surfaces. The amount of the accumulated coolant depended on the coolant-to-mainstream velocity ratio. For low velocity ratios, coolant was found near the suction surface while for higher velocity ratios, it was found near the pressure surface. Zhang et al. (2015) [44] studied phantom cooling of a suction surface by coolant which was intended to cool the endwall. They varied the area of the cooling holes and coolant mass flow rate while keeping the mass flux constant. It was found that the cooling effectiveness on both the suction surface and the endwall increased with increasing hole area up to a particular

![Image](image_url)

Figure 1.3.3 Phantom cooling of suction surface for different velocity ratios [45]
value of the hole diameter. Li et al. (2018) [45] also studied phantom cooling of suction surface on a similar cascade. Their results indicated a higher suction surface phantom cooling for coolant-to-mainstream velocity ratios between 0.4 and 0.6, as shown in fig. 1.3.3. An increase in the coolant mass flow rate was beneficial for the suction surface cooling. However, they found that the pressure surface was not cooled as significantly as the suction surface.

Du et al. (2017) [46,47] numerically studied suction surface phantom cooling due to coolant injection from a slot upstream of the vane passage. They computed changes in cooling performance on changing the upstream slot geometry, coolant slot injection angle with the pitch direction, and mainstream turbulence intensity. A contoured upstream slot provided less phantom cooling than a flat slot. The changes in the slot angle had a considerable impact on cooling performance but the mainstream turbulence did not.

As Nawathe et al. (2021) [41] had considered the combustor coolant injection in their studies, their flow physics was different than that observed in the studies mentioned above. They found that the pressure surface phantom cooling was much more significant than the suction surface phantom cooling (fig. 1.3.4). They also found that while low, the cooling effectiveness values along the suction
surface were still comparable to those in the studies without combustor coolant injection. This further shows the importance of considering multiple coolant injection sources in experiments to accurately capture the cooling performance in the engine.

1.4. Decay of Turbulence through Vane Passage

Freestream turbulence has a direct impact on the vane passage pressure losses, vane surface heat transfer, and injected coolant flows. Blair et al. (1989) [48] found that when the freestream turbulence intensity of the vane passage approach flow is increased, heat transfer in the leading-edge region of the vanes increases substantially and so does the heat transfer in the downstream part of the passage. Similar observations were made by Wang et al. (1999) [49] for a blade cascade. They found that the boundary layer along the suction surface transitioned faster owing to higher turbulence intensity and led to an increase in heat transfer at that surface. They were able to show that the length scales of turbulence also play a role in boundary layer transition. However, they were not able to come up with a correlation between the two. Carullo et al. (2011) [50] were able to show that smaller integral length scales lead to higher heat transfer rates along the surface and were able to corroborate conclusions drawn by earlier studies.

Although it is known that freestream turbulence affects vane passage heat transfer, very few studies have characterized in detail how changes in turbulence affect the cascade flow. As the flow accelerates and turns in a vane passage, the changes in turbulence cannot be predicted using analytical methods. Turbulence in an accelerating flow was first studied by Taylor (1935) [51] by taking measurements in a contraction. He measured the velocity fluctuations in both the streamwise and cross-stream directions for a variety of contraction ratios. He found that, while the velocity fluctuations reduced in the streamwise direction (an effect that can be called ‘decay of turbulence’), velocity fluctuations in the cross-stream direction either increased or decreased with streamwise distance, depending on the contraction geometry and upstream velocity. Dryden and Abbott (1948) [52] also concluded that the turbulence features through a contraction depend on
various factors and cannot be generalized easily. A more comprehensive study of turbulence of flow through contractions was performed by Uberoi (1956) [53]. He showed that the anisotropy of turbulence of a flow through a contraction is high enough that assuming isotropic turbulence for calculating turbulence length scales of the largest eddies will lead to incorrect results. However, for smaller eddies, the flow can still be assumed to be ‘locally’ isotropic and the evaluation of smaller length scales with an isotropic turbulence assumption is valid. He also documented the changes in power spectral distributions before and after the contraction. Similar types of measurements were taken by Radomsky and Thole (1999) [54] and Vicharelli and Eaton (2006) [55] in a turbine vane passage but they did not provide the evolving power spectral distribution inside the vane passage.

Numerous comparisons of computational and experimental results pertaining to vane passage flows are present in the literature. Most of them show that the RANS models do not adequately match their experimental counterparts. Though there is vast literature on this topic, only one representative study by El-Gabry et al. (2018) [56] is discussed here. It is one of the more recent studies that closely resembles the test facility of the current study. It investigated the effects of angle of injection of coolant introduced to cool the endwall of a vane. They documented both the experimental measurements and the computations performed using three RANS turbulence closure models. All numerical results showed a higher endwall cooling effectiveness compared to the experimental results. However, the general trend of the cooling distribution in the streamwise direction was reasonably well-captured by the computations. For similar types of film cooling studies, Acharya et al. (2001) [57], Lynch et al. (2011) [58], and Ravelli and Barigozzi (2017) [59] found turbulent mixing to be underpredicted by the computations, leading to better endwall cooling performance compared to measured results. A comparison of vane passage turbulence features obtained through both experiments and computation using the Reynolds Stress Model (RSM) was done by Radomsky and Thole (2000) [60]. Although the RSM does not make the isotropic turbulence assumption made by eddy viscosity RANS models, it still fails to accurately predict turbulence features in a vane cascade. As the eddy viscosity models are
computationally less expensive than RSM calculations, they are the most widely used turbulence models in industry. Hence, their applicability in predicting freestream turbulence features in a vane passage must be documented, which is the main purpose of one of the studies presented in this thesis.

1.5. Organization of the Thesis

As coolant flows interact with the mainstream and develop complex fluid mechanics, it is imperative to understand such interactions to improve the cooling performance of vane passage surfaces by reducing the required amount of coolant. Mainstream turbulence also plays a major role in the performance of film cooling. Hence, a thorough understanding of evolution of turbulence features in the vane passage is helpful to increase the cooling effectiveness of vane passage surfaces. This chapter has provided the latest understanding of the passage flowfield in the presence of film cooling injection in a first-stage vane passage.

A wind tunnel facility is used to conduct a majority of the results presented in this thesis. The details of this facility are provided in Chapter 2. The extensive measurements recorded in the facility required a variety of instruments. The discussion of these instruments, along with their calibration procedures, is given in Chapter 3. Before conducting any experiments, the test facility must be verified to be imitating the engine conditions correctly. Chapter 4 presents the measurements that qualify that the test facility is operating appropriately.

In Chapter 5, the performance of a newly designed combustor-turbine interface with a modified coolant injection scheme was reported. Aerodynamic and thermal measurements were taken in the passage. These measurements provided a detailed insight to the secondary flow physics and coolant transport. They also helped in understanding how well this injection scheme cools the endwall and vane surfaces, compared to the engine-representative injection scheme used in Alqefl et al. (2021) [37-39] and Nawathe et al. (2021) [40,41].

After the analysis of the results presented in Chapter 5 was completed, it was suspected that the current coolant injection scheme could potentially produce a complete change in the secondary flow system. The coolant flowrates considered
in *Chapter 5* did not show this change. Therefore, aerodynamic measurements for a different range of coolant flowrates were recorded, which are presented in *Chapter 6*. The results in this chapter showed that combustor coolant flowrate establishes which secondary flow features will be dominant in the vane passage. The results presented in *Chapter 5* and *Chapter 6* together are expected to provide gas turbine engineers with ample information to help them design coolant injection schemes that can provide high overall coolant effectiveness and good coolant coverage on the vane passage surfaces.

Understanding the changes in the turbulence features in the vane passage can explain the film cooling effectiveness observed on vane passage surfaces. *Chapter 7* provides detailed measurements that quantify the decay of turbulence in the vane passage. They were conducted in the same wind tunnel facility that was used for the studies presented in the preceding two chapters. Numerical simulations of highly turbulent flows performed using Reynolds-Averaged Navier-Stokes (RANS) turbulence closure models have been shown to not give satisfactory results. To understand where these models fail, a computational study of the vane passage is conducted with RANS models and compared with the experimental measurements. This comparison is expected to help the turbulence modelers to improve existing RANS models for gas turbine applications.

*Chapter 8* summarizes the important results presented in this thesis and their impact on gas turbine applications. It also provides suggestions for future studies that could prove beneficial for further improvement of the efficacy of the film coolant injection technique.
Chapter 2. Experimental Facility Description

2.1. Overview

This chapter will provide the details of the wind tunnel facility that was used to conduct all experimental results presented in this thesis. The facility is located at the Convective Heat Transfer Laboratory at the University of Minnesota. The facility is a result of the combined efforts of many authors and their individual contributions are reported in the following theses: Burd (1998) [61], Oke (2001) [62], Piggush (2005) [63], Erickson (2010) [64], Ayaskanta (2013) [65], Saxena (2015) [66], Alqefl (2016) [67], Alqefl (2019) [68], and Nawathe (2019) [69]. Each author has helped in building or rebuilding of different parts of the test facility to satisfy their experimental conditions. The current state of the facility is shown in fig. 2.1.1. The test facility consists of two main blowers that generate mainstream flow, two auxiliary blowers that generate required turbulence features while adding to the mainstream flowrate, and two coolant blowers that provide the combustor coolant. It also consists of a combustor-turbine interface that simulates coolant injection, a vane cascade that imitates a first-stage, high-pressure vane, and a

![Test facility overview](image-url)
diffuser for the wind tunnel flow to exit in the ambient. Each component of this facility will now be explained. As most of the test facility is similar to the facility described in the author’s master’s thesis, many sections of this chapter are reiterations of the information presented in Nawathe (2019) [69].

2.2. Mainstream Flow Supply

To simulate engine conditions, the Reynolds number of the mainstream must be matched. For a typical engine, this value lies somewhere between 350,000 and 450,000 when calculated based on the vane axial chord length. To generate flowrates that satisfy this requirement, powerful blowers are required. Therefore, the wind tunnel facility utilizes four blowers to generate the mainstream flow. Figure 2.1.1 shows the two main blowers indicated at the entry to the wind tunnel. The green blower is a centrifugal blower that is manufactured by New York Blower. It is a general-purpose blower of type 183 ACF with a power rating of 3.7 kW (5 HP). This blower takes the air from the ambient and its output is fed to an axial blower, which is the second of the two main blowers. This blower has a rated power of 5.6 kW (7.5 HP). It is manufactured by Joy Manufacturing Company and its model number is 29-17-17770AP. The blower can be run up to a frequency of 70 Hz as opposed to a typical 60 Hz. The current experiments use a frequency of 64 Hz as the electrical current drawn at higher frequencies can damage the motor of this blower.

A round-to-square transition duct is located at the exit of the axial blower. A heat exchanger is located at the exit of this duct. It was part of an earlier study and is not utilized for current experiments. A series of screens is present downstream of the heat exchanger to break down the swirl to the flow. Downstream of these screens, a turbulence generator is situated. The design and the working of the turbulence generator is explained in a later subsection. The two auxiliary blowers that feed this generator add to the mainstream flow. Both are manufactured by Dayton Electric Manufacturing Company and are of model 4c330. Each has a power rating of 3.7 kW (5 HP) and together provide about 60% of the flowrate to the mainstream flow. Inlets to all blowers have filters to ensure that the flow
entering the wind tunnel is free of debris. Eliminating debris is important as stray particles can easily break hot-wires of the anemometers and clog the pressure taps on the five-hole probe.

2.3. Turbulence Generator

The turbulence characteristics of the passage inlet flow must match the conditions at the exit of a dry, low-NOx type combustor. The exit flow of this combustor is characterized by high turbulence intensity and large turbulence length scales. As discussed in section 1.4, the mainstream turbulence intensity affects film cooling considerably. Therefore, matching the engine conditions for the wind tunnel flow is crucial. The turbulence generator in the current test facility was originally designed by Erickson (2012) [64] and subsequently modified by Saxena (2015) [66]. The turbulence conditions required in Saxena’s work were the same as the current study and hence, no further modifications were made to the turbulence generator. Only the most relevant details about the turbulence generator are given in this chapter and readers are advised to read the above two theses for a thorough discussion.

![Mainstream Direction](image)

Figure 2.3.1 Turbulence generator (Original drawing from [67])
Figure 2.3.1 shows the turbulence generator. It consists of a large mixing chamber. The main flow (also called the ‘core flow’) enters the chamber from the front wall. It is generated by the two main blowers discussed earlier. It is passed through 12 rectangular slots that are located around the four edges of the front wall. The design of the slots is expected to generate turbulence energy length scales that are in the order of 24-47% of the true chord length of the vanes downstream. Additionally, two separate flows are injected from the auxiliary blowers. These flows travel through several holes in the top, side, and bottom walls. These holes generate jets that are in a direction perpendicular to the mainstream direction, which are called ‘crossflow jets.’ The interaction of the core
flow jets and the crossflow jets leads to turbulence. The geometric details of the slots and the holes on the walls of the mixing chamber are shown in fig. 2.3.2. The turbulence generated due to the jets created by the slots and the holes generates turbulent eddies having the desired length scales and helps in maintaining shear layers that are necessary to preserve the generated turbulence.

2.4. Nozzle

After passing through the turbulence generator, the flow enters a nozzle that enables the change of the cross-sectional area between the turbulence generator upstream and the test facility downstream. A uniform velocity is generated when the flow is contracted in the nozzle. Engine flows also go through an area contraction near the combustor exit. The nozzle in the test facility simulates this contraction. Figure 2.4.1 shows the nozzle. As with the turbulence generator, it was first designed by Erickson (2012) [64] and the latest iteration was made by Saxena (2015) [66]. The original nozzle had a contour that was defined to have the first and second derivatives to the contour curve equal to zero at the entry and the exit of the nozzle. The contour curve and its equation are shown in fig. 2.4.2. Small changes to the contour were made by Saxena as needed for her experiments and the same contour was used for the measurements in this thesis.
2.5. Test Section

Downstream of the nozzle, the flow enters the test section where all measurements are conducted. This section consists of a combustor-turbine interface followed by a vane cascade. It imitates the most downstream region of the combustor liner wall, the region between the combustor exit and the first stage of vanes (also called nozzle guide vanes), and the vane linear cascade. The combustor-turbine interface consists of rows of holes that inject coolant intended to cool the combustor liner and the endwall. The vane cascade has three vanes that make two passages and the endwall is contoured. The top and front views of the test section are shown in fig. 2.5.1 and 2.5.2, respectively.

The test section walls that come in direct contact with the wind tunnel flow are made of acrylic plastic, PVC plastic, or wood to prevent any heat transfer to these surfaces. The acrylic walls provide visual access required to position and traverse the measurement probes. The only exception is a movable piece of the wall that

Figure 2.5.1 Top view of the experimental test section
is made of aluminum. This wall section is used to insert the probe in the test section at different locations (which will be discussed in section 2.6). This wall is located on the opposite side of the contoured endwall. No measurements are taken there.
Hence, although the aluminum will conduct heat, the adverse effects on the recorded measurements will be insignificant.

2.5.1. Combustor-turbine Interface

The mainstream flow in the nozzle has been simulated correctly to match the combustor exit flow. In the engine, the hot gas mixture generated in the combustor can harm the combustor liner as the mixture temperature is much higher than the melting temperature of the liner material. Therefore, coolant is injected to cool the combustor wall. These coolant flows are expected to reach the first stage of vanes and interact with the vane passage flow. Hence, in addition to matching the main flow conditions with the engine, the coolant injection characteristics also need to be matched. The combustor-turbine interface was designed to satisfy this requirement.

Figures 2.5.1 and 2.5.2 show the close-coupled combustor-turbine interface with its neighboring components along with the important dimensions in the facility. The manufacturing of the interface was done in the College of Science and Engineering Machine Shop. It is made of PVC plastic. The geometric uncertainties in the manufacturing of the interfaces, based on the CNC machine used were 0.1 mm in distance and 0.01° in angle. The angle of the interface wall with the mainstream flow direction was designed to be 4.59°. However, due to an error in manufacturing, an angle of 2.6° was achieved after assembling the interface in the test facility. It was believed that this deviation of angle will not have significant impact on the measurements taken downstream in the vane passage. Discussions with Solar Turbines supported this conclusion.

The close-coupled interface has eight coolant rows. The holes of the upstream seven rows have same diameter, angle of injection, spacing and length/diameter ratio. The primary objective of the coolant introduced through these seven rows is to cool the front face of the interface wall which is exposed to the mainstream. The holes of the most downstream row have a different design as this row is expected to especially provide coolant to the endwall. Dimensions of the cooling holes are shown in fig. 2.5.1 and 2.5.2. A separate, metered flow line (not shown), located
behind the test facility, is designed to provide coolant air to the interface. The design of the cooling scheme of this interface was done after studying the results presented in Alqefl et al. (2021) [37-39] and Nawathe et al. (2021) [40,41]. These studies indicated that the combustor coolant can provide significant cooling to the endwall and vane surfaces. Therefore, the injection scheme for combustor coolant was modified to be more evenly distributed in the close-coupled interface. Also, the new design couples the transition duct and the first stage of the vane assembly, eliminating the cavity between the combustor and turbine sections, which is a typical feature of these engines. For the close-coupled interface, such cavity is expected to be upstream of the interface. It is not simulated in the current facility and is not expected to have any impact on the cooling performance in the vane passage.

The coolant supply is metered using a laminar flowmeter upstream of the plenum. The meter is manufactured by Meriam and the model number is 50MC2-4. The flow is provided by a combination of two blowers: one manufactured by Twin City Fans (Model 19N4 TBNA), having a rated power of 5 HP (3.7 kW) and another manufactured by Cincinnati Fans (Model HP-4C22), rated 5 HP (3.7 kW). A speed controller made by MagneTek (Model GDP333) was used to control the frequency of the two blowers. The inlet air to both blowers comes from the ambient.

To distinguish coolant flow from the mainstream, the coolant flow is heated as a way of marking it. To heat the flow, a series of resistance heaters are installed between the exit of the blowers and entry to the laminar flowmeter. They provide a maximum total power of 4.4 kW. The power input can be controlled using a variable autotransformer to achieve and maintain exact coolant air temperature.

2.5.2. Contoured Endwall

The contoured endwall is on the same side of the test facility as the coolant injection holes (also called the 'hub endwall'). Therefore, a majority of the measurements are taken on and near this endwall, making it one of the most important regions in the test section. The endwall (and the vane cascade) was manufactured by Saxena (2015) [66]. It is contoured axisymmetrically, i.e., its
shape changes only in the axial (X) direction and not in the pitchwise (Y) direction. The endwall contour helps in suppressing secondary flows. It was provided by the company sponsor, Solar Turbines Inc., and is shown in fig. 2.5.3. In the streamwise direction, the contour first contracts to accelerate the flow near the endwall and then expands in the latter half of the vane passage. The overall effect is a net acceleration to the flow near endwall from inlet to the exit of the passage as the vane span at the passage inlet is longer than the vane span at the passage exit. The endwall opposite to the hub endwall is called the shroud endwall and is flat in this study. The flat endwall does not imitate the engine geometry and hence, no measurements in this region are recorded.

The contoured endwall is manufactured using medium density fiberboard (MDF). As adiabatic cooling effectiveness is measured on the endwall. This material was chosen due to its low thermal conductivity (0.15 W.m⁻¹.K⁻¹). The fabrication was performed using a CNC machine to have good accuracy. The endwall was fabricated as a single piece with recesses for the three vanes. After machining, the endwall surface was painted to close the pores in the MDF.

Next, the vanes were installed in the recesses on the endwall. A self-adhesive caulk sealant was used to seal the small gaps between the endwall and the vanes creating round fillets of 3 mm radius between these components. Then, the vane-endwall assembly was installed in the test section. The assembly was supported with a wooden frame. This frame was supported with a structure made from

![Figure 2.5.3 Contoured endwall in reference to the vanes](image-url)

Figure 2.5.3 Contoured endwall in reference to the vanes [67]
Unistrut® beams, which were bolted to the test section support structure (a workbench).

2.5.3. Vane Cascade

The vane cascade used in the current study simulates first-stage, high-pressure nozzle guide vanes. This cascade has been used previously in the following studies: Saxena (2015) [66], Alqefl (2016) [67], Alqefl (2019) [68], and Nawathe (2019) [69]. The vane cascade is a stationary and linear cascade that sufficiently imitates vanes in the engine.

The purpose of the nozzle guide vane in the engine is to accelerate the flow coming out of the combustor and turn it to align it for the entry to the first stage rotors. The test facility cascade is made up of three vanes that create two passages as shown in fig. 2.5.2. The leading edge of the top vane (vane 1) and the leading edge of the bottom vane (vane 3) are at the same pitchwise (Y) location as the top and bottom walls of the combustor-turbine region, respectively. The geometric details of the vane cascade are listed in table 2.5.1.

Although the vanes in the engine are designed to be transonic, the vanes in the test facility are modified to match the low Mach number (<0.2) experimental conditions. The cascade vanes are 4.91 times the engine vane size. This increase helps in matching the Reynolds number of the engine flow with the wind tunnel flow. The larger scale of the test facility also provides good measurement resolution, which helps in resolving the passage flowfield. The scaling is based upon the vane pitch (distance between the leading edges of successive vanes) of the cascade and the engine. The vanes have moderate loading as indicated by a Zweifel coefficient of 0.91. Due to the constraints of the test facility, the vane inlet span (Z) in the test is shorter than the equivalent inlet span in the engine. Due to the contoured endwall, the span of the vane changes (highest at the passage inlet and lowest at the highest contoured part of the endwall) throughout the passage. Two bleed slots are located at the top and the bottom wall of the test facility just upstream of the leading edges of vanes 1 and 3. The slots are present for the whole span and their thickness is adjustable. The flow is leaked or bled from these
slots to remove the boundary layers on the top and bottom walls upstream of the cascade. These boundary layers will not be present in the engine as the vanes in the engine are of an annular arrangement. The flow is bled in a way to have periodic flow between two vane passages (section 4.2 reports the vane passage periodicity demonstration).

Table 2.5.1 Cascade vane geometric specifications

<table>
<thead>
<tr>
<th>Specification</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Scale Factor (with respect to actual engine size)</td>
<td>4.91</td>
</tr>
<tr>
<td>Vane True Chord Length (C)</td>
<td>36.54 cm</td>
</tr>
<tr>
<td>Vane Axial Chord Length (C_{ax})</td>
<td>20.53 cm</td>
</tr>
<tr>
<td>Cascade Pitch (P)</td>
<td>32.25 cm</td>
</tr>
<tr>
<td>Vane Inlet Span (S_i)</td>
<td>33.72 cm</td>
</tr>
<tr>
<td>Vane Exit Span (S_e)</td>
<td>31.3 cm</td>
</tr>
<tr>
<td>Vane Aspect Ratio, Inlet (S_i/C)</td>
<td>0.922</td>
</tr>
<tr>
<td>Vane Aspect Ratio, Outlet (S_e/C)</td>
<td>0.857</td>
</tr>
<tr>
<td>Space-Chord ratio (P/C)</td>
<td>0.883</td>
</tr>
<tr>
<td>Vane Inlet Angle (α, angle between cascade centerline axis and camber line at leading edge)</td>
<td>0°</td>
</tr>
<tr>
<td>Vane Outlet Angle (β, angle between cascade centerline and camber line at trailing edge)</td>
<td>72.33°</td>
</tr>
<tr>
<td>Inlet Passage Height (H_i)</td>
<td>64.50 cm</td>
</tr>
<tr>
<td>Vane-Endwall Fillet Radius</td>
<td>0.3 cm</td>
</tr>
</tbody>
</table>

The vanes are manufactured by Saxena (2015) [66] based on the profile provided by Solar Turbines Inc. For easy machining and to have low thermal conductivity (0.19 W.m^{-1}.K^{-1}), vanes are made of acrylic plastic. The accurate manufacturing of the vanes was achieved using horizontal and vertical CNC milling machines along with the Mastercam software. Forty-five thin slabs that have the vane shape were manufactured for the three vanes (see fig. 2.5.4). Fifteen slabs constitute to a single vane. Seven of these slabs had multiple holes of diameter 0.508 mm drilled into them in the span direction up to the mid-span (when measured from the contoured endwall side) throughout the vane. Then, for each
of these resulting channels, a hole was drilled perpendicular to them through either the pressure or the suction surface (see fig. 2.5.5). The correct alignment of the perpendicular holes was achieved by using a 3-D printed jig. The exact locations of these channels and the perpendicular holes are shown in fig. 2.5.6. When the wind tunnel is operating, each of these channels will be pressurized according to the static pressure at the vane surface location where they were drilled. Hence, when the pressure from each of these holes is measured, the pressure distribution profile along the pressure and suction surfaces of the vanes can be plotted.

2.5.4. Test Section Access

To take measurements at different locations, there must be access to the test section at any location in the vane cascade. For this wind tunnel, a part of the flat endwall side of the test facility can be removed. This consists of four rigid panels and one panel with a moveable hole through which the probe can be inserted.
the facility as shown in fig. 2.5.7. The dimensions of the rigid panels were decided in a way that enables the moveable hole panel to be attached in any location where the rigid panels can be attached. This makes it possible to take measurements at any axial position downstream of the nozzle and up to the vane passage exit. All edges of the panels have grooves of the same size. These slide into the grooves made on the support frame that is bolted to the test section table. Toggle clamps are used to hold the panels in place. While running the wind tunnel, tape segments are put on the borders of each of the panels to prevent the leakage from the small gaps left between the grooves. As mentioned before, all panels and walls are made of acrylic plastic to avoid heat transfer and to get visual access to the vane passage.
Figure 2.5.6 Static pressure tap locations along the vane surfaces [66]
The moveable-hole panel was designed by Piggush (2005) [63]. It is shown in fig. 2.5.8. The panel consists of a smaller circular disk that is installed inside a larger circular disk in a non-concentric fashion. These disks can rotate using the ball bearing rings attached to the outer side of their circumferences. The smaller disk has a hole along the inner side of its circumference through which a probe can be inserted. The diameter of the smaller disk is equal to the sum of the radius of the larger disk and the hole diameter. Due to this design, this hole can travel
anywhere in the circular area covered by the larger disk allowing measurements to be taken at various locations without stopping the wind tunnel and without any major leakage. The structure of the panel is made of aluminum for rigidity while the two disks are made of acrylic plastic.

2.6. Exit Flow Management

2.6.1. Tailboards

As the wind tunnel is an open-return type, the flow in the cascade must be continuously released into the ambient. It must be diffused carefully before its exit from the wind tunnel because that reduces the pressure rise requirement on the inlet blowers. Tailboards, which are present downstream of the cascade, were designed by Alqefl (2016) [67] to direct the passage flow through the diffuser. They are shown in fig. 2.5.7. The two tailboards are attached to the trailing edges of
vanes 1 (top vane) and 3 (bottom vane), respectively. The diffuser wall is designed to seamlessly blend with the contoured endwall to create a single smooth surface. On the opposite side, a flat diffuser wall was designed to merge with the flat endwall that is along the vane passage. The tailboards are made of acrylic plastic sheets of thickness 0.635 cm (0.25 in). The exact profiles of the tailboards are found by modifying their locations while measuring the static pressure distribution in the two vane passages. When the distribution in the two passages match, the correct profile of the tailboards is achieved. Then, the tailboards are held in place using multiple clamps attached to the diffuser wall.

2.6.2. Diffuser

After the flow is directed by the tailboards, diffuser channels are needed to diffuse the flow. Usually, a long and straight single-channel diffuser is the best choice for diffusion. However, due to the space constraints in the room in the vertical direction, a shorter diffuser was designed by Alqefl (2016) [67] based on a
two-dimensional computational analysis. Figure 2.6.1 shows this diffuser. The outlet-to-inlet area ratio of the diffuser is 2.9. The diffuser vanes are made of acrylic plastic sheets with a thickness of 0.238 cm (0.094 in). The diffuser walls are made by joining two thin wooden sheets together. Grooves are made into the upper sheet that have the thickness of the diffuser vanes. This allows the diffuser vanes to be inserted and held between the diffuser walls. Due to the thin diffuser vanes, this arrangement was not enough to hold the vanes in place. While running the wind tunnel, the generated vibrations caused the diffuser vanes to pull out of the grooves. Therefore, thin, yet sturdy, steel reinforcements were designed and attached to the upstream end of the diffuser vanes. This decreased the amplitude of the vibrations of the diffuser vanes and prevented them from coming out of the grooves.
Chapter 3. Instrumentation

This chapter will provide details of various instruments used throughout this thesis. It will give brief explanations of the working principle of the instruments, which quantities were measured, and how these instruments were calibrated. The majority of the instrument details are the same as presented in Nawathe (2019) [69] and hence, many parts of this chapter are replications of the information presented in that reference. Details of each instrument will now be discussed.

3.1. Thermocouple

Two types of thermal measurements were needed for the experiments presented in this thesis. The first were to describe the coolant transport in the vane passage to understand the mixing of coolant with the mainstream and the second were to measure surface cooling effectiveness of the vane passage surfaces. Both measurements were taken by thermocouples. The working principle of a thermocouple is based on the Seebeck effect. It states that if two different metals are joined and the temperature of their junction is made to be different than the temperature at the other end of these metals, an electromagnetic field is generated. This field gives rise to a small potential difference that can be related to the difference between the two temperatures described above. Although the magnitude of the potential difference is low, it is highly accurate and can be used to measure the temperature at the junction.

Type E (chromel-constantan) thermocouples were used in the current experiments. They can be used in the temperature range of 3-1150 K. All experiments in this thesis are conducted at temperatures near ambient (295-318 K). Hence, Type E thermocouple are suitable for them. The two ends of a thermocouple are called ‘hot’ and ‘cold’ junctions, respectively. While the hot junction is exposed to the location where a temperature measurement is to be taken, the cold junction is kept in an ice bath as a reference. As the temperature measurements are taken for up to 100 hours continuously and over a number of weeks, using an ice bath was impractical. Instead, electronic compensation
provided by the data acquisition unit (DAQ), Agilent 34970A, was applied to the cold junction. It was derived based on the data provided by National Institute of Standards and Technology (NIST) as reported in Burns et al. (1993) [70]. An ice bath was also used to measure some sample temperatures and these measurements were compared with temperatures measured using the DAQ compensation. The difference between them was found to be less than 0.024 °C. This value is well within the uncertainty limits required by the experiments and hence, the compensation by the DAQ is considered suitable for recording temperatures.

3.2. Inclined Manometer

The core principle of an inclined manometer is based on the principle that if a pressure force is applied on an incompressible fluid, its displacement can be directly related with the applied pressure. A standard manometer consists of two vertical tubes that are connected at the bottom (usually in a U-shape) and contain a fluid. When the pressures at the top ends of the tubes are the same, the fluid levels in both tubes are equal. When different pressures are applied to the two top ends, a net pressure force is generated, which displaces the fluid. This gives rise to a difference in the levels of the fluid in the two tubes ($\Delta y$). This difference can be related to the difference between the pressures applied at the top of the tubes ($\Delta P$), as follows:

$$\Delta P = \rho \cdot g \cdot \Delta y$$  \hspace{1cm} (3.1)

Where $\rho$ is the density of the fluid in the tubes and $g$ is the acceleration due to gravity.

The current experiments use an incline-type manometer, shown in fig. 3.2.1. Due to its inclined nature, it is able to provide significant fluid displacement for small pressure differences, which results in exact measurements. The manometer used in these studies was manufactured by Dwyer Co. Model 246. The fluid in the tube is Dwyer red gage oil with a specific gravity of 0.8. The scale on the manometer is adjusted based on this specific gravity to provide not the displacement in terms of ‘inches of the oil column’ but rather in terms of ‘inches of
water column.' The range of this manometer is from 0-6 inches of water column (0-1494 Pa at ambient temperature). The resolution is equal to 0.01 inches of water column (2.5 Pa at ambient temperature), as reported by the manufacturer.

None of the pressure measurements recorded in this thesis are absolute pressures. The reference pressure used for all measurements is the pressure inside the wind tunnel at specific locations along the vane surfaces. Hence, changes in the ambient pressure do not impact the quantities that are derived from the pressure measurements. The inclined manometer is used for two purposes: (i) to calibrate other instruments and (ii) to monitor the coolant flowrate being injected in the test section.

3.3. Pressure Transducer

Although the manometers have high accuracy, the measurements need to be recorded manually. Therefore, it is not practical when a large volume of data must be collected. However, they can be used to calibrate other, less-accurate pressure measuring instruments that can record measurements automatically. Pressure transducers are one of the instruments capable of such process. They are faster and more precise than manometers. They can send signals to a data acquisition
system that, in turn, sends these signals to a computer where they can be stored and processed.

A pressure transducer contains a diaphragm made of magnetically permeable stainless steel. This diaphragm deforms under the application of pressure and that leads to a change in the magnetic reluctance of the stainless steel. The change in reluctance also changes the inductance of a coil embedded in the transducer. The coil is connected in a circuit and due to the change in its inductance, the voltage in the circuit also changes. As this change of voltage occurs due to the applied pressure, a correlation between the voltage and the pressure can be found. The voltage signal is sent to a computer via a DAQ where it is converted into voltage using a MATLAB program. Pressure transducers made by Validyne Engineering Corp. are used in this study. Their model number is DP15. The five-hole pressure tube probe (explained in the following section) measures five pressure differences simultaneously. Hence, five transducers were used in this study.

The calibration of these transducers is performed using the inclined manometer. A uniform velocity jet is generated in a calibration jet facility, and the dynamic pressure of this flow is measured by the manometer. At the same time, a pressure transducer is connected in parallel, and the voltage generated across it is recorded using a DAQ. The pressure drops in both lines (one going to the manometer and the other going to the transducer) are maintained equal. This process is repeated in a way that the dynamic pressures span across the entire operating range of the transducers. Once this procedure is over, a set of pressures and corresponding voltages are achieved. These are plotted to produce a correlation between the pressure and the voltage. Detailed descriptions of the calibration setup and process are presented in Alqefl (2019) [68] and are not repeated here. The correlations for all transducers are given below. The pressure, \( P \), is in inches of water column while the voltage, \( V \), is in Volts. The suffixes denote the number on the labels of each transducer.

\[
P_1 = 0.0045 * V_1^2 + 1.3258 * V_1 + 0.0116 \tag{3.2}
\]
\[
P_2 = -0.0003 * V_2^2 + 0.5369 * V_2 + 0.0141 \tag{3.3}
\]
\[
P_3 = -0.0005 * V_3^2 + 0.5394 * V_3 + 0.0103 \tag{3.4}
\]
\[ P_4 = 0.0001 \times V_4^2 + 0.5235 \times V_4 + 0.2162 \quad (3.5) \]
\[ P_5 = -0.0014 \times V_5^2 + 0.5421 \times V_5 + 0.0191 \quad (3.6) \]

The calibration range for the pressures was 0-6 inches of water column (0-1494 Pa at ambient temperature) as higher pressures are not expected in the test facility. The corresponding voltages of the transducers are 0-5 Volts for the transducer with the correlation of equation 3.2 and 0-10 Volts for the rest of them, equations 3.3-3.6. The coefficients of the second-degree terms in the above equations are close to zero signifying a near-linear relationship between pressure and voltage. This was expected and therefore confirms that the diaphragms of all transducers are working correctly.

### 3.4. Five-hole Pressure Tube

For understanding the flow patterns in the vane passage, both the magnitudes and the directions of the velocity vectors must be measured. A five-hole pressure tube, also known as a five-hole probe, can be used to obtain such information. The five-hole probe used in current studies is shown in fig. 3.4.1. As the name suggests, it consists of five holes at its tip. The center hole (1) can measure the total pressure of the flow only in the situations where the flow is normal to the cross-section of the hole. Most frequently, this will not be the case, a calibration (discussed later in this section) is performed to find a correction factor that can covert the ‘measured pressure’ into the ‘total pressure.’ The difference between the pressures measured at holes (2) and (3) is used to calculate the pitch angle of the flow. This is the angle that the probe makes with the X-Z plane. Similarly, pressures measured at holes (4) and (5) give the angle of the velocity vector with the X-Y plane, which is known as the ‘yaw’ angle. The co-ordinate system is shown in fig. 3.4.2 for reference. The total pressure and two angles of the flow are enough to calculate the three components of the velocity vector. Before explaining the process of calculating the velocity components, the calibration procedure is discussed.
Calibration Procedure

Calibrating the five-hole probe is necessary as all five pressures measured by the probe are needed to define each of the three quantities to be extracted (total pressure and two flow angles). Therefore, the calibration procedure involves creating a simulation where flow comes toward the probe from a variety of different angles and the pressures measured at each of the five holes are recorded simultaneously. Then, this set of measurements is used to generate contour fits for several non-dimensionalized pressure coefficients that are used to calculate velocity components.

The report by Treaster and Yocum (1979) [71] was used to design the calibration procedure of the five-hole probe. Out of the two techniques given in this reference, 'nulling' and 'non-nulling,' the latter was chosen. In this method, the flow used for calibration is always imparted with a fixed orientation and the probe angles are changed for each measurement. In this way, with reference to the probe tip,
the flow appears to come from different pitch and yaw angles. A calibration rig was 3-D printed to generate this flow simulation by Alqefl (2016) [67]. It is shown in fig. 3.4.3. The constant air flow is generated by a leaf blower. The velocity of this flow is about 52 m.s⁻¹, which is approximately equal to the flow velocity at the exit of the vane passage in the wind tunnel. A nozzle is attached at the end of the blower to generate a uniform velocity profile. Figure 3.4.3 shows the location of the tip of the probe and the direction of the incoming flow from the leaf blower. If the probe is aligned exactly at the location shown, it is also aligned at the center of airflow coming out of the blower. This ensures that the flow reaching the probe tip is the same for all calibration measurements. Therefore, any changes in pressures detected by the five-hole probe are due to the angle of the probe.

The calibration rig provides a large range of possible pitch and yaw angles of the probe. Each can be changed independently of the other. The pitch angle can be changed in increments of 5° while the yaw angle can be changed in increments of 1°. The calibration was performed for a range of ± 35° with increments of 5° for both yaw and pitch. Two additional yaw angles, +2° and -2°, were added in the calibration measurement matrix to achieve better accuracy to the curve fit in the low-angle region. Therefore, for each of the 15 pitch angle positions, 17 yaw angle positions were tested. This led to a total of 255 unique probe positions for which the five pressures were recorded. As the Tygon tubes connected to the five-hole probe respond slowly to pressure changes, a 30-second waiting period was spent
before starting the recording of measurements at each probe location. After the waiting period, measurements were recorded continuously for 20 seconds and then averaged. The probe sends the pressure signals to the transducers (discussed in section 3.3) and the voltages generated by the transducers are sent to the DAQ. These voltages can then be converted back into pressures using the correlations (equations 3.2 to 3.6).

Once the pressures are recorded, they are converted into various pressure coefficients (\(C_p\)) defined by Treaster and Yocum (1979) [71]. These are listed below:

\[
C_{p_{\text{total}}} = \frac{P_1 - P_{\text{total}}}{P_1 - P_{\text{ave}}} \tag{3.7}
\]

\[
C_{p_{\text{pitch}}} = \frac{P_2 - P_3}{P_1 - P_{\text{ave}}} \tag{3.8}
\]

\[
C_{p_{\text{yaw}}} = \frac{P_4 - P_5}{P_1 - P_{\text{ave}}} \tag{3.9}
\]

\[
C_{p_{\text{static}}} = \frac{P_{\text{ave}} - P_{\text{static}}}{P_1 - P_{\text{ave}}} \tag{3.10}
\]

Where

\[
P_{\text{ave}} = \frac{P_2 + P_3 + P_4 + P_5}{4} \tag{3.11}
\]
The subscripts on the pressure terms denote the following:

1-5: Pressure measured at the holes of the five-hole probe (see fig. 3.4.1 for the nomenclature of holes)

total: Relating to the total pressure of the flow

static: Relating to the static pressure of the flow

pitch: Relating to the pitch angle of the flow

yaw: Relating to the yaw angle of the flow

The denominators in equations 3.7-3.10 have an average pressure term, $P_{ave}$. This is a normalizing term and is defined by Treaster and Yocum as shown in equation 3.11. However, this definition of the average pressure results in a singularity for the pressure coefficients at higher flow angles (> 30°). Therefore, Hall and Povey (2017) [72] redefined the normalizing term as given below:

$$P_{ave} = \frac{1}{2} \{ \min(P_2, P_3) + \min(P_4, P_5) \}$$  \hspace{1cm} (3.12)

As higher flow angles were expected in the wind tunnel flow, the above definition was chosen to define the average pressure term.

Once the pressure coefficients are obtained for all 255 probe orientations, they were used to generate various contour fits using interpolation performed in MATLAB using the ‘lowess’ (locally weighted scatterplot smoothing) method. There were four contours created and all of them were three-dimensional. Their details are listed in Table 3.4.1. The ‘Fitting Parameters’ are calculated using the measured pressures. The ‘Evaluated Parameters’ are calculated through the ‘Fitted Parameters’ using the appropriate equations from the expressions (3.7)-(3.10) and (3.12). As the ‘lowess’ method does not generate a visualization of the fitted contours, these contours cannot be shown. Initially, a single contour fit was generated for the entire range of flow angles. But this fit created significant errors (>2.5°) for higher flow angles. Therefore, two separate sets of contour fits were generated. The first set was for the range of probe orientations where both the pitch and yaw angles were less than 20°. The second set was generated for the
remainder of the probe orientations. This helped in reducing the uncertainty in actual measurements that is due to error in the calibration contour fit.

Table 3.4.1 Details of the contour fits

<table>
<thead>
<tr>
<th>Fitted Parameter</th>
<th>Fitting Parameters</th>
<th>Evaluated Parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pitch</td>
<td>$C_{p_{\text{pitch}}}, C_{p_{\text{yaw}}}$</td>
<td>Pitch</td>
</tr>
<tr>
<td>Yaw</td>
<td>$C_{p_{\text{pitch}}}, C_{p_{\text{yaw}}}$</td>
<td>Yaw</td>
</tr>
<tr>
<td>$C_{p_{\text{total}}}$</td>
<td>$C_{p_{\text{pitch}}}, C_{p_{\text{yaw}}}$</td>
<td>$P_{\text{total}}$</td>
</tr>
<tr>
<td>$C_{p_{\text{static}}}$</td>
<td>$C_{p_{\text{pitch}}}, C_{p_{\text{yaw}}}$</td>
<td>$P_{\text{static}}$</td>
</tr>
</tbody>
</table>

It should be noted that although each of the holes on the five-hole probe is identified to be related to pitch, yaw, or total pressure, the pressures measured at all five holes are needed to define each of these quantities. Hence, if the accuracy of a measurement of even one of these three quantities is in question, it is highly likely that all other quantities also have significant errors associated with them.

Velocity Components Calculation Procedure

When measurements are taken in the wind tunnel using the five-hole, the five voltages corresponding to the five holes of the probe are recorded. These are converted to pressures ($P_1$ through $P_5$). Then, the pressure coefficients are calculated and finally, the ‘Evaluated Parameters’ in Table 3.4.1 are found. The next step is to calculate the absolute velocity magnitude, $V$, using the following formula:

$$V = \sqrt{\frac{2(P_{\text{total}}-P_{\text{static}})}{\rho}}$$  \hspace{1cm} (3.13)

Where $\rho$ is the density of air in the wind tunnel. After this, the velocity components can be evaluated using absolute velocity magnitude ($V$), pitch angle ($\alpha$), and yaw angle ($\gamma$) by equations derived in Treaster and Yocum (1979) [71]. They are listed below:

$$V_X = V \times \cos(\gamma) \times \cos(\alpha)$$  \hspace{1cm} (3.14)

$$V_Y = V \times \cos(\gamma) \times \sin(\alpha)$$  \hspace{1cm} (3.15)

$$V_Z = V \times \sin(\gamma)$$  \hspace{1cm} (3.16)
Where the subscripts X, Y, and Z denote the directions of the velocity components (see fig. 3.4.2 for the co-ordinate system). The uncertainty in absolute velocity magnitude is found to be 2.5%. The uncertainty in the pitch and yaw angles was found to be 2°. Both uncertainties are defined with a 95% confidence limit. The detailed calculations of uncertainties are provided in Appendix A.

3.5. Laminar Flowmeter

In the current test facility, a mainstream flow and a coolant flow is present. The mainstream flow has a constant flowrate while the coolant flowrate is changed for different cases. To accurately achieve the required coolant flowrate, it must be measured with a sophisticated flowmeter. Therefore, laminar flowmeters are used to meter the coolant flow.

In a laminar flowmeter, the flow is passed through several thin tubes that are attached in parallel. Due to the small diameters of these tubes, the flow becomes laminar. This flow is a Hagen-Poiseuille flow and hence, has an exact analytical solution. Therefore, the velocities evaluated using the pressure drop across the laminar flowmeter are highly accurate. As the cross-sectional area of the tube is known, the calculated volume flowrate is also very accurate. Besides the cross-sectional area and the pressure drop across the flowmeter, only the viscosity of the fluid is needed. This can be calculated with ease as the temperature of the fluid is known. A disadvantage of using a laminar flowmeter is that it causes a significant drop in pressure in the coolant flow supply line. This means that blowers that provide high flowrates at high pressure rise will be needed. The blowers described in section 2.5.1 satisfy this requirement.

A laminar flowmeter manufactured by Meriam (Model 50MC2-4) was used in this study. The pressure drop across the flowmeter is measured using two pressure taps provided at the entry and exit of the flowmeter. Manometers are used to read these pressure drops. To convert the measured pressure into flowrate, the manufacturer has provided the following equation:

\[ Q = \left( -0.0692834 \Delta P^2 + 50.0651 \Delta P \right) \frac{\mu_{std}}{\mu_f} \]  

(3.17)
Here, the pressure drop, \( \Delta P \), is in inches of water column while the volumetric flowrate, \( Q \), is in cubic feet per minute. The term \( \mu_{std} \) relates to the dynamic viscosity of air at 21 °C while \( \mu_f \) is the viscosity of air passing through the laminar flowmeter. The manufacturer has reported an uncertainty of 0.72% with 95% confidence interval for the measured volumetric flowrate.

The coolant flowrates are decided based upon the coolant-to-mainstream mass flowrate ratios (MFR). The combustor coolant MFR is defined below:

\[
(MFR)_{\text{combustor}} = \frac{Q_{\text{combustor}}}{V_0 \ast A_m}
\]  

(3.18)

As the densities of the coolant and mainstream are approximately equal, the mass flowrate ratio is equal to the volume flowrate ratio. The MFR values are first decided based on the objectives of the study. Then, the mainstream velocity, \( V_0 \), is measured far upstream of the vane passage. The cross-section, \( A_m \), at the location of this velocity measurement is calculated. Then, using equation 3.18, the required combustor coolant flowrate, \( Q_{\text{combustor}} \), is found.

### 3.6. Hot-wire Anemometry

Some of the tests presented in this thesis require characterization of turbulence of the wind tunnel flow at various locations. High-frequency velocity measurements are required to perform this characterization. Hence, the slow-responding, five-hole probe is not suitable and a hot-wire anemometer was used instead.

There are three types of hot-wire anemometers: (i) constant-temperature type, (ii) constant-current type, and (iii) constant-voltage type. This study uses the constant-temperature type anemometer. For this anemometer, a very thin wire attached at the end of the hot-wire probe is used to sense the velocity of the flow to be measured. It is connected to an electrical circuit and its temperature is kept constant by maintaining a constant electrical resistance. As the velocity of the flow changes, the convective heat transfer coefficient of the flow changes. This change can also lead to a change in the wire temperature. However, the heat transfer rate between the flow and the hot wire is changed such that the hot wire temperature will not change. Therefore, the rate of energy produced by the hot wire (called
resistance heating) changes according to the convective heat transfer coefficient. The equation of the rate of energy or power is shown below:

\[ P = \frac{(Q)}{R} \]

The resistance of the hot wire is constant as it is to be maintained at a constant temperature. Therefore, the current in the circuit is changed according to changes in the convective heat transfer coefficient. The current is changed by changing the voltage applied to the circuit. Therefore, a correlation can be found between the velocity of the flow, \( u \) (which changes the convective heat transfer coefficient), and the voltage of the circuit, \( V \). As shown by King (1914) [73], the form of this correlation is as follows:

\[ u^n = A \times V^2 + B \]  

The terms \( n \), \( A \), and \( B \) are curve-fit constants. They are different for each hot-wire and are evaluated by calibrating the hot-wire probe. The value of \( n \) typically lies between 0.35 and 0.5.

The hot-wire anemometer used in this study is manufactured by TSI Inc. It is a Constant-Temperature Anemometer (CTA) of Model 1750. The hot-wire probe is a Standard Boundary Layer Probe manufactured by TSI Inc. and its model number is 1218. It is shown in fig. 3.6.1. The boundary layer probe is useful when measurements near a wall are needed. The temperature of the hot-wire is maintained at 250 °C. This value is suggested by the manufacturer as it optimizes the sensitivity and the lifespan of the hot-wire.

![Model 1218 Standard Boundary Layer Probe](image)
Overview of Calibration Procedure

The hot-wire calibration setup is shown in fig. 3.6.2. As it is discussed in detail in Nawathe (2019) [69], only a brief description is provided here. The setup was built based on the one documented in Wilson (1970) [75]. The calibration jet device is used to generate a circular jet of air having a uniform velocity profile. The hot-wire probe senses this flow and sends the corresponding signal to the CTA anemometer which, in turn, sends the voltage to the DAQ. The voltage magnitude is then transferred to and stored in the computer. Just upstream of the exit of the jet from the calibration jet device, a static pressure tap is located. The pressure measured at this tap is sent to a manometer. As the calibration jet exits to the atmosphere with minimum pressure loss between the static pressure tap and the exit, the total pressure of the jet is the same as the ambient pressure. Therefore,

Figure 3.6.2 Hot-wire calibration setup [66]
when the other end of the manometer is kept open to the ambient, the reading shown on the manometer is the dynamic pressure of the jet. A thermocouple inserted in the calibration jet device (shown in fig. 3.6.2) measures the temperature of the flow. This temperature is used to calculate the density of the air in the jet. Using the dynamic pressure and the density of the air jet, the exit velocity of the jet (which is sensed by the hot-wire probe) is calculated. In this way, the jet velocity (measured by the manometer) and the corresponding voltage (measured by the hot-wire anemometer) are collected simultaneously. These two sets of parameters can be used to generate a curve that has an equation with the same form as equation 3.20.

The inclined manometer could measure a maximum dynamic pressure of 5.3 inches of water column (1319 Pa). This corresponded to a maximum velocity of 47 m.s$^{-1}$. Although slightly higher velocities ($\sim$54 m.s$^{-1}$) are realized at some locations,
in the wind tunnel, the hot-wire probe was only calibrated up to 47 m.s\(^{-1}\) and extrapolated data from the resulting curve-fit equation was used for higher velocities. Due to the smoothness of the fitted curve, this extrapolation was not expected to increase uncertainty of the measurement. The range of dynamic pressures considered for calibration was 0-5.3 inches of water column (0-1319 Pa) with a measurement taken at an increment of 0.1 inches of water column (25 Pa). For each measurement point, the voltage data were collected at 2 kHz frequency for 25 seconds, and averaged. The plot of the jet velocity against the measured voltage is shown in fig. 3.6.3. The data points reflect the measured values while the solid line is the fitted curve. The curve-fit equation is also shown in the figure. The fitting is performed using the least-square method. It agrees well with the measured values. Velocity magnitudes were found to have an uncertainty of 3.8\% within a 95\% confidence interval. Details of the uncertainty analysis are provided in Appendix A.
Chapter 4. Test Section Qualification

Before the experiments begin, it is important to ensure that the wind tunnel facility correctly represents engine conditions. Hence, a set of qualification tests must be performed before the actual measurements are made. These tests are listed below:

(a) Matching the Reynolds number of the flow in the test section with the engine flow Reynolds number
(b) Verifying that the flows passing through the two passages of the vane cascade are matched
(c) Validating that the velocity distribution profile and the turbulence features of the flow approaching the vane cascade are similar to the engine flow
(d) Testing the adiabaticity of the endwall and vane surfaces and confirming that the thermal profile of the approach flow represents the engine thermal profile

Each of these tests will now be discussed in detail.

4.1. Reynolds Number of the Vane Cascade Flow

Typically, the vane passage inlet Reynolds number of the engine flow, based on the true chord length, is higher than 350,000. At this Reynolds number, the viscous effects are negligible. Hence, a Reynolds number above this value must be achieved in the test facility for establishing proper test conditions. The Reynolds number is calculated using the following expression:

\[ \text{Inlet } Re_{chord} = \frac{\rho \times V \times C}{\mu} \]  

(4.1)

Where, \( \rho \) and \( \mu \) are the density and dynamic viscosity of the wind tunnel air and \( C \) is the true chord length. The term \( V \) represents the average velocity at the passage inlet. It was measured using the five hole-probe measurements taken at the passage inlet (Passage inlet corresponds to Plane 2 in Chapters 5 and 6). The Reynolds number was found to be 362,000. Therefore, the test facility flow satisfies the Reynolds number requirement.
4.2. Flow Periodicity between Vane Passages

The vane cascade in the current test facility consists of three vanes that form two passages. It is supposed to represent the annular ring present in the engine. To verify that the flow through the passage matches the flow through an annular ring, flow periodicity must be achieved between the two vane passages. The first step of the periodicity tests is to record static pressures at the surfaces of these vanes at different axial locations throughout the vane passage. Then, the pressures are converted into non-dimensional pressure coefficients, and they are plotted with respect to their axial positions to develop a pressure profile. As there are two vane passages, the profiles of the two suction surfaces and the two pressure surfaces must match for flow periodicity. The flow through the vane passages is adjusted by moving the downstream tailboards (discussed in section 2.6) until a match between the pressure profiles is achieved.

The static pressures on the vane surfaces are measured using several pressure taps located at the midspan of the vane surfaces. These were discussed in section 2.5.3 and are shown in fig. 2.5.6. These pressure taps are connected to a pneumatic switchboard using Tygon tubes. Each pressure tap has an individual switch. By turning on the switch of a particular pressure tap, the pressure in the corresponding Tygon tube is applied to one end of a pressure transducer. The other end of the pressure transducer is connected to a reference pressure. This reference is one of the static pressure taps in the wind tunnel. A reference inside the wind tunnel is used for two reasons: (i) The effect of changes in the ambient pressure will not affect the measurements and (ii) Any transient changes in the wind tunnel flow will be felt by both the reference pressure and the pressure to be measured, ensuring that the effects of these changes will be canceled out.

The pressure profiles are measured for four vane surfaces that form the two vane passages: (a) suction surface of Vane 1, (b) suction surface of Vane 2, (c) pressure surface of Vane 2, and (b) pressure surface of Vane 3. The vane nomenclature is shown in fig. 4.2.1. The non-dimensionalized pressure coefficient $C_p$, at an axial location, $X/C_{ax}$, is defined as follows:
\[ C_p(X/Cax) = \frac{P_{\text{static}}(X/Cax) - P_{\text{stag}}}{0.5 \cdot \rho \cdot U_{\text{exit}}^2} \] (4.2)

Where \( P_{\text{static}} \) is the measured static pressure, \( P_{\text{stag}} \) is the stagnation pressure at the leading edge of the vanes, and \( U_{\text{exit}} \) is the velocity at the passage exit measured using a pitot static tube.

The measured pressure profiles of the four surfaces are shown in fig. 4.2.1. They generally match well. Near \( X/Cax = 0.6 \), a slight mismatch is observed. But this local mismatch is not very significant as the rest of the profiles are matching.

4.3. Approach Flow Velocity and Turbulence Features

As the turbulence influences the transport of the coolant and the generation of secondary flows, the approach flow characteristics must imitate the flow conditions in the engine. The approach flow plane is located 16.63 cm upstream of the passage inlet \( (X/Cax = -0.81) \); shown in fig. 4.3.1). This location was chosen

![Figure 4.2.1 Static pressure profile of vane surfaces](image-url)
because most of the coolant has been injected in the main flow and the location is
enough upstream of the vane passage to not experience any effect of the curvature
of the passage. Also, as there are no sharp gradients in the geometry near the
approach plane location, no separation is expected.

4.3.1. Bulk Flow Features

The instantaneous velocity of the flow is measured using a hot-wire
anemometer. The measurements are taken at the approach flow plane at 14
pitchwise rows (Y) and each row had 19 spanwise measurement locations (Z). The
measurements are recorded for one whole pitch distance (P) and for up to 95% of
the span distance (S) when measured from the contoured endwall. The
measurement grid is shown in fig. 4.3.2. At each location, measurements are taken
for 25 seconds at a frequency of 2 kHz. The Agilent 24465A data acquisition
system is used to record the data and send it to a computer. Three bulk flow
properties are calculated based on the measured instantaneous velocities: (i)
mean velocity, $\overline{U}(Y,Z)$, (ii) root-mean-square (RMS) fluctuations of the
instantaneous velocity, \( u'(Y,Z) \), and (iii) turbulence intensity, \( Tu(Y,Z) \) (in percent). Their expressions are given below:

\[
\bar{U}(Y,Z) = \frac{1}{N} \sum_{1}^{N} u(Y,Z,t) \tag{4.3}
\]

\[
u'(Y,Z) = \frac{1}{N} \sum_{1}^{N} \left( u(Y,Z,t) - \bar{U}(Y,Z) \right)^2 \tag{4.4}
\]

\[
Tu(Y,Z) = \frac{u'(Y,Z)}{\bar{U}(Y,Z)} \times 100 \tag{4.5}
\]

Where

\( u(Y,Z,t) \): The instantaneous velocity measured by the hot-wire anemometer at location \( (Y,Z) \) at a time instance \( t \).

\( N \): Number of samples collected over the recording time (25 seconds for current measurements)

Figure 4.3.2: Velocity measurement locations for the approach flow characterization plane
The mean velocity contours are shown in fig. 4.3.3. The core velocities are close to 13 m.s\(^{-1}\). The velocities near the contoured endwall are elevated due to the injection of combustor coolant and reach values as high as 20 m.s\(^{-1}\). Away from the contoured endwall, an overall pitchwise and spanwise uniformity in the velocities is observed. This confirms that the approach flow velocity distribution is satisfactory to conduct the experiments. Similarly, RMS velocity fluctuations near the contoured endwall are up to 2 m.s\(^{-1}\) while in the mainstream they are around 1.2 m.s\(^{-1}\), as shown in fig. 4.3.4. Finally, contours for turbulence intensity, which is the ratio of the RMS fluctuations to the mean velocity, are shown in fig. 4.3.5. The average turbulence intensity is observed to be around 10\%, which is representative of engine turbulence intensity. Overall, the bulk flow properties satisfy the qualification requirements.

Figure 4.3.3 Mean velocity contours at the approach flow characterization plane
4.3.2. Turbulence Features

The bulk flow turbulence intensity is not enough to sufficiently characterize the turbulence. The size of the eddies in the flow establishes the coolant transport in the engine, which affects the vane passage surface cooling performance. In low-NOx combustors, large eddies are generated that take long distances to dissipate and these must be correctly simulated by the turbulence generator in the test facility (discussed in section 2.3). The results that verify the characteristics of the generated turbulence are discussed in this subsection.

The turbulence kinetic energy (TKE) spectrum can be plotted using the high-frequency velocity measurements taken by the hot-wire anemometer. The frequency of recording measurements was 2 kHz, which is sufficient to capture the eddies up to the inertial subrange of the turbulence spectrum.

Figure 4.3.4 RMS velocity fluctuations at the approach flow characterization plane
For the spectral analysis, it is assumed that ‘Taylor’s Hypothesis of Frozen Turbulence’ holds. It assumes that while an eddy is passing across the probe location (which will take a finite time as the eddy has a finite size), it does not change its size or shape. In other words, it is frozen as it passes the probe. If this hypothesis holds, it allows taking measurements at a single point in space over a short timespan to characterize the turbulence. This assumption holds when the change in the mean velocity magnitude of the flow due to turbulence is less than half of the mean velocity magnitude. Based on the contours of bulk flow properties for the current test facility, this assumption is valid for the current test section.

The first step toward plotting the TKE spectrum is to calculate the instantaneous velocity fluctuations ($u'(t)$) compared to the mean velocity ($\bar{U}$) using the following equation at every measurement location:

\[
\text{Figure 4.3.5 Turbulence intensity distribution at the approach flow characterization plane}
\]
\[ u'(t) = u(t) - \bar{U} \]  \hspace{1cm} (4.6)

Where

\[ \bar{U} = \frac{1}{N} \sum_{0}^{N} u(t) \]  \hspace{1cm} (4.7)

These velocity fluctuations are in the time domain. The next step is to convert them to the frequency domain. This conversion is performed using a predefined function in MATLAB that uses a Fast Fourier Transform (FFT). The transformation is done using the following equation:

\[ Q_k = \frac{1}{\sqrt{N}} \left( \sum_{n=1}^{N} u'(t)_k \cdot e^{i\left(\frac{2\pi n}{N}\right)_k} \right) \]  \hspace{1cm} (4.8)

Where,

\( Q_k \): the transformed array from FFT,
\( k \) = index of \( Q_k \) array (\( k = 1, 2, \ldots, N/2 \))
\( n \) = index of \( u'(t) \) (\( n = 1, 2, 3, \ldots, N \))

The transformed array ‘\( Q_k \)’ is a complex number with its real part describing the amplitude of the velocity fluctuation at index ‘\( k \)’ and the imaginary part describing the phase difference between the fluctuation at index ‘\( k \)’ and the velocity fluctuation at index 1 (i.e., at time, \( t = 0 \)). Based on this transformation, the energy density function, \( E(w) \), can be evaluated as follows:

\[ E(w) = \frac{Q_{real}[f]^2 + Q_{imag}[f]^2}{f_s/2} \]  \hspace{1cm} (4.9)

Where ‘\( f_s \)’ is the sampling frequency and ‘\( f \)’ is the fluctuation frequency defined as:

\[ f = \frac{f_s k}{N} \]  \hspace{1cm} (4.10)

The energy density function can then be plotted against the fluctuation frequency to obtain the TKE spectrum (also called power spectral distribution). Sizes of different eddies, which represent various turbulence length scales, can be evaluated using this plot.
Power spectral distributions at multiple locations were plotted and compared and it was found that the spectrum at the mid-pitch mid-span location provides a good representation of the power spectral distribution in the flow. It is shown in fig. 4.3.6. At frequencies lower than 10 Hz, it is quite possible that the fluctuations recorded were due to some unsteadiness present in the wind tunnel, rather than due to turbulence. Hence, although the data recorded for frequencies below 10 Hz are plotted, they may not correctly represent turbulent kinetic energy at those frequencies. A longer record would reduce this low-frequency variability, but more accurate characterization of this portion of the spectrum was considered to be unnecessary for the objectives of the qualification test. The turbulence length scales, calculated using the TKE spectrum, are now discussed.

**Integral Length Scale, $\Lambda$**

The size of the largest turbulent eddies are characterized by the integral length scale. These eddies occur at low frequencies and are represented by the flat region on the power spectral distribution. They are also called energy containing eddies due to the high TKE associated with them, as seen on the spectral distribution plots. Hinze (1975) [76] gave the following expression to calculate this length scale:

$$\Lambda = \frac{\bar{U}}{4} \lim_{\omega \to 0} \frac{E(f)}{u'^2}$$

(4.11)

The limit in the above expression indicates that the TKE at extremely low frequency should be used for calculating the integral length scale which is in line with the fact that the larger eddies have smaller frequencies. Hence, only measurements below 1 Hz are used to get a value for the TKE. As the number of data points corresponding to frequencies less than 1 Hz are very low and the wind tunnel flow may have low-frequency disturbances, as discussed before, the calculation of the integral length scale involves significant uncertainty. Also, the integral length scale is dependent on the flow geometry and hence, would be different at different locations on the approach plane. Based on multiple power spectral distributions that were taken from near the center of the plane to near any of the vane passage surfaces, the integral length scale lies between 2.5 cm and 5 cm.
As the measurements at low frequencies had possible uncertainties, the integral length scale was also calculated using another method. This method does not require the low-frequency TKE but uses extrapolation of data points using an empirical expression. It is known as the ‘von Karman Law.’ The expression is shown below:
The integral length scale is varied in the above equation until the resulting plot of this expression (black plotline in fig. 4.3.6) matches the mid-frequency region (100-300 Hz) of the measured TKE spectrum (yellow plotline in fig. 4.3.6). The mid-frequency regions of the measured and empirical plots are matched because the data in this region are most reliable as they have enough data points (unlike the low-frequency region) and have little possibility of aliasing error (unlike the high-frequency region). Substituting integral length scale values of around 3 cm in equation 4.12 resulted in a fit. The plot in fig. 4.3.6 is calculated by substituting an integral length scale of 2.8 cm in the equation. Based on both the measured values and the von Karman law, it is concluded that, in general, the integral length scale for the approach flow will lie in the range of 2.5 cm to 5 cm. Note that an integral length scale was not found for every single measurement location. Hence, there is a possibility that, at some locations, the integral length scale value may be outside the reported range.

**Energy Length Scale, \( L_u \)**

The energy length scale is defined using the turbulence kinetic energy, \( k \), and the turbulence dissipation rate, \( \varepsilon \). Although it lacks a physical representation in the flow, the energy scale is very useful in comparing turbulence features generated in different experimental facilities. The first step in evaluating this scale is to define the kinetic energy dissipation, \( \varepsilon \). It is defined as the rate at which the turbulence kinetic energy is converted into the internal energy by way of viscous dissipation. On the power spectral distribution, the inertial subrange, which is characterized by the drop in the energy after the flat region (>50 Hz), is used to calculate the dissipation. For this region, the Kolmogorov’s 5/3rd law holds. It states that ‘in the inertial subrange, the turbulence kinetic energy is proportional to the \(-5/3\)rd power of the wave number.’ Due to a linear relationship between the wave number and frequency of the eddies \( (\omega) \), the same relation holds between the energy \( (E(\omega)) \)
and the eddy frequency \((w)\) in the frequency domain. The expression for the Kolmogorov relation is given below:

\[
E(f) = \frac{18}{55} \cdot A \cdot \varepsilon^{2/3} \cdot f^{-5/3} \cdot \left(\frac{2 \cdot \pi}{U}\right)^{-2/3}
\]  (4.13)

As the power spectral distribution is plotted on a log-log scale, a natural log is taken on both sides. This leads to following expression:

\[
\log(E(f)) = -\frac{5}{3} \log(f) + \log\left(\frac{18}{55} \cdot \left(\frac{2 \cdot \pi}{U}\right)^{-2/3} \cdot A \cdot \varepsilon^{2/3}\right)
\]  (4.14)

Where ‘A’ is a constant and equal to 1.62, according to Ames and Moffat (1990) [77].

Equation 4.14 is equivalent to the equation of a line of the form, \(y = m \cdot x + c\). Hence, ideally, the last term of this equation is constant for any combination of \(E(f)\) and \(f\). This equation is used to evaluate dissipation \(\varepsilon\), which is part of the last term, using the spectral distribution. In reality, the calculated \(\varepsilon\) values have small variations depending on which combination of \(E(f)\) and \(f\) was chosen from the spectrum. But, for the approach plane, this variation was found to be insignificant. Therefore, one such combination of \(E(f)\) and \(f\) was chosen and the corresponding value of dissipation was calculated to be 20.7 m².s⁻³. After this, the energy length scale \(L_u\) is evaluated using the following expression:

\[
L_u = 1.5 \cdot \frac{\overline{u'}^3}{\varepsilon}
\]  (4.15)

The value of the energy length scale was found to be 8 cm. The hot-wire probe used in these tests can only measure the major component of velocity. At the approach plane, the flow direction is mostly axial. Hence, the subscript \(u\) is added in \(L_u\), which states that this length scale value corresponds to the axial direction only. At the approach plane, the turbulence is expected to be isotropic (same in all directions) but as it cannot be verified with a single wire hot-wire probe, it is not claimed that the energy length scale has the same value in all principal directions.

**Taylor Microscale, \(\lambda\)**

The larger eddies dissipate into smaller and smaller eddies and the smallest eddies dissipate into thermal energy due to the effect of viscosity. The smallest
eddy size is characterized by the Kolmogorov length scale. However, many researchers opt to not report this scale as there is uncertainty attached to its calculation from the measured data. Instead, another length scale is chosen for reporting turbulence features. This is known as the Taylor microscale, \( \lambda \). This length scale represents eddies that are smaller than the energy containing eddies but larger than the Kolmogorov-scale eddies. Due to their small size, they are found to be dependent on the dissipation rate and the fluid kinematic viscosity (\( \nu \)). Hinze (1975) [76] provided the following expression to calculate the Taylor microscale:

\[
\lambda = \sqrt{\frac{15 \ast \nu \ast u'^2}{\varepsilon}}
\]  

(4.16)

The small-scale turbulence features are not dependent on flow geometry. Hence, this length scale practically has the same value irrespective of measurement location. It was found to be 0.35 cm at the approach plane.

All major quantities evaluated in this section are summarized in table 4.3.1. However, most of the values corresponding to turbulence features do not give a clear indication about whether the generated turbulence is representative of that in the engine. Therefore, these features are compared with values measured in a combustor simulator designed by Ames (1997) [78]. This reference is known to provide a very good imitation of the turbulence at the combustor exit. Also, it was used to build the turbulence generator in the current facility. As the dimensions of the two facilities are different, Ames suggests that the height of the simulator exit, \( H_c \), can be used to scale the results. The comparison between results obtained from the two setups is shown in table 4.3.2. Although Ames’ setup had slightly higher inlet velocity and turbulence intensity, the non-dimensionalized values between the two setups are found to match well. This serves as a validation that the turbulence generator of the current facility produces engine-representative turbulence.
Table 4.3.1 Approach flow characteristics

<table>
<thead>
<tr>
<th>Bulk Flow Characteristics</th>
<th>Turbulence Characteristics</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mean Velocity, $\bar{U}$ (m.s$^{-1}$)</td>
<td>Integral Length Scale, $\Lambda$ (cm)</td>
</tr>
<tr>
<td>12.8</td>
<td>2.5-5</td>
</tr>
<tr>
<td>Velocity Fluctuations, $U_{rms}$ (m.s$^{-1}$)</td>
<td>Energy Length Scale, $L_u$ (cm)</td>
</tr>
<tr>
<td>1.25</td>
<td>8.0</td>
</tr>
<tr>
<td>Turbulence Intensity, $T_u$ (%)</td>
<td>Taylor Microscale, $\lambda$ (cm)</td>
</tr>
<tr>
<td>9.77</td>
<td>0.35</td>
</tr>
<tr>
<td>Cascade Reynolds Number, $Re_{chord}$</td>
<td>Dissipation, $\varepsilon$ (m$^2$.s$^{-3}$)</td>
</tr>
<tr>
<td>362,000</td>
<td>20.7</td>
</tr>
</tbody>
</table>

Table 4.3.2 Comparison of turbulence characteristics

<table>
<thead>
<tr>
<th>Facility</th>
<th>$H_C$ (cm)</th>
<th>$\bar{U}$ (m.s$^{-1}$)</th>
<th>$U_{rms}$ (m.s$^{-1}$)</th>
<th>$T_u$ (%)</th>
<th>$\Lambda$ (cm)</th>
<th>$L_u$ (cm)</th>
<th>$\Lambda/H_C$ (%)</th>
<th>$L_u/H_C$ (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ames’</td>
<td>42.54</td>
<td>19.3</td>
<td>1.6</td>
<td>8.3</td>
<td>2.08</td>
<td>4.34</td>
<td>4.89</td>
<td>10.20</td>
</tr>
<tr>
<td>Current</td>
<td>92.7</td>
<td>12.8</td>
<td>1.2</td>
<td>9.8</td>
<td>2.8</td>
<td>8.0</td>
<td>3.02</td>
<td>8.63</td>
</tr>
</tbody>
</table>

4.4. Thermal Qualification Results

4.4.1. Adiabaticity of Vane Passage Surfaces

Some of the thermal measurements recorded in this thesis are used to calculate adiabatic cooling effectiveness of vane passage surfaces (section 5.2.3 will provide the reason behind choosing to measure the ‘adiabatic’ effectiveness). Therefore, it should be verified that there is no heat transfer to these surfaces. The adiabaticity of the contoured endwall has previously been tested by Saxena (2015) [66]. It was done by taking temperature measurements very close to the endwall and verifying that the temperature gradient is zero. It confirmed that the contoured endwall is adiabatic. The details of this test are not repeated here.

As phantom cooling measurements on vane surfaces are taken as part of one of the studies in this thesis, the adiabaticity of the vanes must be established. Similar to the contoured endwall, temperature measurements at different locations near the vane surfaces are recorded. It was found that up to 1 mm away from the vane surfaces, the temperature was uniform. In other words, the temperature gradient was practically zero. The measurement technique of traversing thermocouple helps ensure that the temperature measurements are taken within...
1 mm of the vane surface. Therefore, the vane surfaces can be considered adiabatic.

4.4.2. Approach Flow Thermal Profile

Based on the pitchwise (Y) uniformity of the velocity profiles, it can be said that the coolant is injected uniformly through all cooling holes. Hence, a thermal profile at the approach plane (X/Cax = -0.81) was not recorded. The first measurement plane of the study reported in Chapter 5 is located at X/Cax = -0.104, which is just downstream of the last injection row. The thermal contours plotted for this plane (see Plane 1 in section 5.3) show that the coolant thickness along the endwall does not change in the pitch direction implying pitchwise uniform coolant presence just before the flow is entering the vane passage. Therefore, it is concluded that the thermal profile of the passage inlet flow match that of the engine.
Chapter 5. Performance of the Close-coupled Combustor-turbine Interface Geometry

5.1. Motivation

Experiments conducted by Alqefl et al. (2021) [37-39] and Nawathe et al. (2021) [40,41] have shown that the coolant injected to cool the combustor helps in cooling the endwall and vane surfaces. This information was used to design a new combustor-turbine interface, the close-coupled interface, that can provide better utilization of various coolant flows injected upstream of the first-stage vanes. The design has already been discussed in section 2.5. This chapter will discuss various measurements taken in the test facility to record the performance of this new combustor-turbine interface. These measurements will describe, in detail, the secondary flow physics, the coolant transport, and cooling effectiveness on vane passage surfaces. The goal of this study is to understand whether this new coolant injection scheme has the potential to reduce the overall coolant requirement in the gas turbine engine.

5.2. Experimental Procedure

5.2.1. Experimental Cases

Four coolant-to-mainstream mass flow rate ratios (MFR) were considered for this study. Details of these cases are shown in table 5.2.1. Each case name has the form CXX where C indicates combustor coolant and the two numbers following indicate the MFR. For example, the Case C56 indicates that the combustor coolant is injected at the MFR of 5.6%. The cases were chosen in a way that they can be related with at least one case from one of the previous studies on this facility (Alqefl et al. (2021) [37-39] and Nawathe et al. (2021) [40,41]), which had an engine-representative combustor-turbine interface. These comparison cases are also listed in the table. The lowest MFR is 5.2% which relates to the lowest MFR taken in the previous studies. It is believed that going below this number may not be
possible due to constraints on cooling imposed by the combustor section. The highest MFR considered was 6.6%. In previous studies, the case with a net MFR of 6.58% was found to give the best performance among all cases. As the combustor-turbine interface considered in the current study is expected to give more coolant coverage than that of the engine-representative combustor-turbine interface cases, cases using the present interface that have higher MFR than 6.6% are not needed. The experimental cases of this study still cover the most important range of coolant flow rates.

Table 5.2.1 Experimental cases

<table>
<thead>
<tr>
<th>Case</th>
<th>Total MFR (%)</th>
<th>Comparable Cases from Previous Studies</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Alqefl et al. (2021) [37-39]</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Effusion MFR (%) Louver MFR (%) Film MFR (%)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>C52 5.2 4.66 0.37 0 - - -</td>
</tr>
<tr>
<td></td>
<td></td>
<td>C56 5.6 - - - 4.66 0.92 0</td>
</tr>
<tr>
<td></td>
<td></td>
<td>C61 6.1 4.66 0.37 1 4.66 1.43 0</td>
</tr>
<tr>
<td></td>
<td></td>
<td>C66 6.6 4.66 0.37 1.5 4.66 0.92 1</td>
</tr>
</tbody>
</table>

5.2.2. Aerodynamic Measurements Procedure

To understand the vane passage flow physics, it is important to characterize the interactions of mainstream and coolant flows. Detailed aerodynamic measurements, which consist of velocity magnitudes and flow angles at various locations throughout the vane passage, are required. These data can then be compared with the thermal measurements that provide information on coolant transport and cooling of vane passage surfaces.
Figure 5.2.1 shows the isometric view of the vane passage (with one of the vanes removed for better visibility). It is the co-ordinate system used for all studies in this thesis. The aerodynamic data are collected at four axial locations (X). For each axial location, the measurements are recorded in a Y-Z plane for the whole pitch (Y, distance from vane pressure surface to suction surface) and for distances up to 50% of the vane span (Z) from the contoured endwall. Each is referred to as a ‘plane,’ henceforth. These planes are shown in fig. 5.2.2. Additionally, a measurement plane along the throat of the passage (Plane 6) was considered. It has an angle of 64.25° with the axial (horizontal) direction. This plane is located approximately at the throat of the passage. For each plane, 13 to 20 pitch locations are chosen for measurement, depending on the location of the plane. They typically go from the pressure surface to the suction surface. These are referred to as ‘rows.’ In each row, 21 span points that spread from the contoured endwall to
the mid-span were chosen for measurements. The exact measurement locations are shown in fig. 5.2.3. The ordinate (Y) is non-dimensionalized by the pitch distance at the passage inlet (Pi) while the abscissa (Z) is non-dimensionalized by the span distance at the passage inlet (Si). The inlet span and pitch are identified in fig. 5.2.1.

The five-hole probe is used to take these measurements. This probe does not give accurate results for near-wall flows because the pressure recorded at the flanking hole of the probe that is closest to the wall is incorrect due to proximity to the wall. The distance from the surface at which this ‘near-wall’ effect is not present, depends on the design of the probe (Treaster and Yocum (1979) [71]). This distance was calculated, and all measurements were recorded at least this required distance away from the endwall and vane surfaces. For Planes 2 and 3, a limitation of access to the test facility prevented the collection of measurements near the suction surface. Therefore, only a fraction of the plane was available for data recording, as shown in fig. 5.2.3. Also, Plane 2 measurements near the

![Diagram of measurement planes](image-url)

Figure 5.2.2 Axial locations of aerodynamic measurement planes. X/Cax values are indicated for each plane near the suction surface.
pressure surface are in the vicinity of the leading edge. The stagnation region at the leading edge produces a highly unsteady flow. Previous studies on this vane cascade have shown that the flow angles in this region fall well outside the calibration range of the five-hole probe and hence, the recorded measurements will not be accurate. Therefore, Plane 2 measurements that are close to the leading edge are not recorded.

As described in Chapter 1, the vane passage flow physics consists of several secondary flow structures. These are called secondary flows because they deviate from the ideal mainstream flow direction. Therefore, each measured velocity vector can be thought of as consisting of two velocity vectors: (i) the ideal streamline vector at that location and (ii) the deviation (or secondary flow) vector from the ideal streamline vector. In other words, if one were to subtract an ideal streamline vector from the measured velocity vector, the resulting vector is the deviation from the ideal flow and hence, a secondary flow vector.

To begin the process of calculating the secondary flow vector, the first step is to define an ideal streamline vector. For the current studies, an ideal flow is defined as mainstream flow that is unaffected by coolant injection. Based on previous measurements in this vane cascade, it is known that the coolant flow does not reach beyond the mid-span when measured from the contoured endwall side. Therefore, for each row of measurements taken at a particular pitch location, the mid-span measurement can be considered the ideal streamline for that pitch location. To evaluate the secondary flows for all locations at this pitch, this mid-span velocity vector will be subtracted from all measurements at this pitch. The process is repeated for all pitch locations for all planes for all cases. The velocity component equations (3.14-3.16) presented in section 3.4 are now updated to define the secondary flow velocity component equations as follows:

\[
V_{X,SF} = V \cdot \cos(\alpha - \alpha_s) \cdot \cos(\gamma - \gamma_s) \quad (5.1)
\]

\[
V_{Y,SF} = V \cdot \cos(\gamma - \gamma_s) \cdot \sin(\alpha - \alpha_s) \quad (5.2)
\]

\[
V_{Z,SF} = V \cdot \sin(\gamma - \gamma_s) \quad (5.3)
\]

Where the subscript \(SF\) denotes 'secondary flow,' and the subscript \(s\) relates to the ideal streamline. As mentioned above, the values of \(\alpha_s\) and \(\gamma_s\) will be different for
each row of measurements. These secondary flow velocity components are used to plot the vorticity contours, secondary flow vectors, and secondary flow streamlines.

5.2.3. Thermal Measurements Procedure

Two types of thermal measurements are recorded for this study. The first type are the temperature measurements taken in the vane passage to document the coolant concentration through the passage. The second type are the temperature measurements taken at the three vane passage surfaces to study the cooling effectiveness of the coolant injection through the close-coupled, combustor-turbine interface geometry. Both measurements are recorded using a Type-E
thermocouple, but their physical significance is different. They will be discussed in separate subsections.

The fluid used for both the mainstream flow and the coolant flow is air. Therefore, the coolant flow needs a marker to be specifically identified from the mainstream. This marking is achieved by heating the coolant flow by \( \approx 14 \, ^\circ \text{C} \) above the temperature of the mainstream. This means that higher temperature equates to better cooling effectiveness. As the coolant is heated only to be marked, no heat transfer measurements are taken. The purpose of this study is to understand the mixing of coolant flow with the mainstream flow and how the cooling effectiveness is decreased by this mixing, which does not require heat transfer measurements.

**Coolant Concentration Measurements**

To understand how coolant concentration changes in the vane passage, temperature measurements are recorded at multiple axial locations, as done with aerodynamic measurements, so that the secondary flow features obtained from the aerodynamic measurements can be compared with the coolant concentration contours obtained from the thermal measurements. The same axial locations as for the aerodynamic measurements are chosen for the coolant concentration measurements. One additional location, called Plane 1, was chosen for thermal measurements. Figure 5.2.4 gives the details of the measurement planes. The number of pitch rows for each plane ranged between 14 and 22 depending on the plane location. Near the suction surface, the measurements in the spanwise direction were taken as far as 32\% of the span length measured from the contoured endwall. For the pressure surface, this distance was increased to 50\% of the span length. These values were chosen based on the highest distances from the contoured endwall over which coolant was present in previous studies. The measurement locations for each plane are shown in fig. 5.2.5.

The temperature measured by the thermocouple is not the static (or thermodynamic) temperature because of the dynamic heating, thermal diffusion, viscous dissipation, and conduction effects near the thermocouple tip. Therefore, the temperature measured by the thermocouple is called the 'recovery temperature,' denoted by \( T_{re} \). To convert the recovery temperature to static
temperature \( T_{\text{static}} \), it first must be converted to stagnation temperature \( T_{\text{stag}} \) of the air flow. Stickney (1955) [79] defined a Recovery Ratio, \( R \), as follows:

\[
R = \frac{T_{re}}{T_{stag}}
\]

Both temperatures in the above equation are in Kelvin. The recovery ratio is highly dependent on the Mach number (M) of the flow. In the current facility, the highest Mach number measured is about 0.19 at an ambient temperature of 300 K. Stickney explains that the changes in recovery and stagnation temperatures for Mach numbers below 0.2 are negligible. At the peak Mach number, the value of the recovery ratio is equal to 0.997, which is very close to unity. For most of the passage, the Mach number is well below 0.2 and hence, the value of recovery ratio is above 0.997. Therefore, for this study, the ‘recovery temperature’ and ‘stagnation temperature’ are essentially the same.

Figure 5.2.4 Axial locations of thermal measurement planes. X/Cax values are indicated for each plane near the suction surface.
Once the stagnation temperatures were obtained, the static temperatures were found using the isentropic compressible flow table for air. The static temperatures were calculated for maximum velocities measured at all the planes and are documented in table 5.2.2. The differences between static and stagnation temperatures (ΔT) are shown in the last column. For Mach numbers above 0.1, this difference becomes significant. So, the temperature measured by the thermocouple cannot precisely be considered to always be equal to the static temperature.

Figure 5.2.5 Thermal measurement locations
Table 5.2.2 Difference in the static and stagnation temperatures at different velocities

<table>
<thead>
<tr>
<th>Maximum Velocity at Plane</th>
<th>Velocity Magnitude m.s(^{-1})</th>
<th>M (T_{stag}/T_{stat})</th>
<th>(T_{stag}) (T_{stat}) (\Delta T)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Plane 2</td>
<td>18</td>
<td>0.05</td>
<td>0.9995</td>
</tr>
<tr>
<td>Plane 3</td>
<td>30</td>
<td>0.09</td>
<td>0.9984</td>
</tr>
<tr>
<td>Plane 4/5</td>
<td>56</td>
<td>0.16</td>
<td>0.9949</td>
</tr>
<tr>
<td>Plane 6</td>
<td>65</td>
<td>0.19</td>
<td>0.9936</td>
</tr>
</tbody>
</table>

Thermal measurements indicate the cooling potential of the coolant at the location of measurement. The recorded values are converted to a non-dimensional parameter in a way that they indicate the loss of this cooling potential as the coolant moves through the passage. This non-dimensional parameter is called the recovery temperature coefficient, \(\theta\). It is defined as the ratio of the cooling potential at a particular location to the cooling potential of the coolant at the location of injection. It is expressed as follows:

\[
\theta = \frac{T_{(Y,Z)} - T_{\infty}}{T_C - T_{\infty}}
\]  
(5.2)

Where \(T_{(Y,Z)}\) is the recovery temperature measured at location \((Y, Z)\) on a particular plane, \(T_{\infty}\) is the mainstream temperature and \(T_C\) is the coolant temperature at injection. All three values are recorded for each measurement location and the denominator is maintained close to \(\sim 14 \, ^{\circ}\text{C}\). As discussed in the preceding paragraph, the static temperature is not equal to the recovery temperature for high-velocity regions and therefore, the thermal plots in this chapter will not always represent the static temperature, which is the actual (thermodynamic) temperature of the flow. Nonetheless, the objective of having these plots is to understand the coolant mixing due to coolant-mainstream interactions and show how it is affected by changes in coolant flowrate. Therefore, although the recovery temperature coefficients do not represent the exact coolant concentrations, they are more than sufficient to provide valuable insights in coolant transport in the passage. The uncertainty in the measured values of \(\theta\) is 0.016 within a 95% confidence interval (see Appendix A for details).
Surface Effectiveness Measurements

Once the coolant transport is understood, its effect on cooling of the vane surfaces and endwall must be documented. Therefore, temperature measurements are recorded on these surfaces using the same thermocouples used for the coolant concentration measurements. As these surfaces are made of materials with low thermal conductivities, the measured temperature is the adiabatic temperature of those surfaces. As mentioned above, heat transfer to the surface (which will be present in the engine) is not recorded in this study. However, Goldstein (1971) [80] explained that this heat transfer coefficient is relatively close to the heat transfer coefficient between the surface and the mainstream without coolant injection. It means that this value is primarily a function of the mainstream turbulent boundary layer transport and can be found with relative ease. Hence, while performing experiments involving vane passage surface cooling, the important parameter of interest is the adiabatic film cooling effectiveness.

The surface measurement locations used in Nawathe et al. (2021) [40,41] were found to have sufficient resolution to capture the relevant details. They are shown in fig. 5.2.6. These were located at 16 axial locations along the suction surface and endwall while at 14 axial locations (stations) along the pressure surface. For the endwall, measurement locations along 14 out of the 16 stations span the entire pitch (Y) while the remaining stations only have measurement locations near the suction surface, as this region required higher spatial resolution. For the vane surfaces, all rows have 18 measurement locations that cover up to 50% of the local span distance away from the contoured endwall. Table 5.2.3 lists the relevant details of the stations and fig. 5.2.6 shows the station numbers. Some axial locations are the same as the planes used for flowfield measurements. They are identified in table 5.2.3.
Figure 5.2.6 Surface measurement locations
The thermocouple measurements are converted to the non-dimensionalized adiabatic surface effectiveness, $\eta$. It is defined as follows:

$$\eta = \frac{T_{(X,Y,Z)} - T_{\infty}}{T_C - T_{\infty}} \quad (5.3)$$

where $T_{(X,Y,Z)}$ is the recovery temperature measured at location $(X,Y,Z)$ on a particular plane, $T_{\infty}$ is the mainstream temperature and $T_C$ is the coolant temperature at injection. The surface effectiveness is defined similarly to the recovery temperature coefficient, $\theta$, but its significance is different. A value of unity suggests maximum possible coolant coverage while a value of zero suggests no presence of coolant. The uncertainty in surface effectiveness values is 0.016 within a 95% confidence interval (see Appendix A for details).

### 5.3. Vane Passage Flowfield Measurements

This section discusses the aerodynamic as well as coolant concentration measurements recorded in the vane passage. These results are used to explain the cooling performance on the vane passage surface, as will be discussed in the following section.
Plane 1 ($X/Cax = -0.104$)

Plane 1 is located upstream of the vane passage and downstream of the last row of coolant holes. Only thermal measurements were taken on this plane from the contoured endwall to 32% of the local span ($Z/Si$). The thermal contours for this plane are shown in fig. 5.3.1 for the four MFR values. This plane is used to verify that the thermal profile entering the passage is correct. For all MFR cases, a thick coolant layer can be seen near the contoured endwall. For all cases, the coolant layer thickness and the near-wall coolant temperatures are uniform in the
pitchwise (Y/Pl) direction. This confirms that there are no irregularities in the flow entering the passage.

For previous studies (Alqefl et al. (2021) [37-39] and Nawathe et al. (2021) [40,41]), a coolant blob was seen near the suction surface. It was thought to be the effect of two phenomena: the coolant redistribution occurring in the cavity present between the combustor and turbine sections and the coolant “responding to” the pressure distribution in the curved passage downstream. As the cavity present in the previous studies is no longer present for this study, this coolant blob has decreased considerably in size. There is still some coolant redistribution due to passage curvature, but it is minor.

As more coolant is introduced in the passage, one might expect it to project farther in the spanwise (Z/Si) direction into the mainstream. For Plane 1, the coolant layer increases very slightly as the coolant flowrate is increased. The downstream planes show evolving effects of higher coolant flow rates by way of mainstream penetration; but only a hint of this is seen at Plane 1.

**Plane 2 (X/Cax = -0.02, Passage Inlet Plane)**

The aerodynamic and thermal contours for Plane 2 are shown in fig. 5.3.2 through 5.3.5. This plane is located just upstream of the passage inlet and shows the coolant distribution as it enters the passage. The mainstream velocity is not significantly affected by the coolant flowrate as seen in the velocity contours (fig. 5.3.2). Velocities are highest near the suction surface due to the shape of the suction side of the vane and decrease toward the pressure surface. The velocity magnitudes near the endwall show changes with coolant flowrate. These will be discussed along with the Plane 3 velocity contours in the next subsection as they will provide a better explanation of this phenomenon. The mean vorticity on this plane is negligible. This confirms that the flow entering the passage does not bring secondary flows with it and is essentially the same as the combustor exit flow.
Figure 5.3.2 Plane 2: Absolute velocity contours
Figure 5.3.3 Plane 2: Vorticity contours with secondary flow vectors
Figure 5.3.4 Plane 2: Thermal contours
Figure 5.3.5 Plane 2: Thermal contours with secondary flow streamlines
For Plane 2, only, the actual velocity vectors are plotted, as the secondary flow vectors were negligible in magnitude. The vectors along the endwall are much flatter than the vectors in the mainstream. This is an effect of the coolant injected upstream of the passage, which is providing resistance to flow turning due to the shape of the vanes downstream of Plane 2. As the coolant flow rate increases, the vectors become flatter as this resistance rises for cases with higher coolant momentum. The coolant concentration along the endwall is pitchwise (Y) uniform for all cases with higher values of the recovery temperature for higher MFR cases. Along the pressure surface (Y/Pi = 0), the coolant spreads slightly toward the midspan, as observed in the thermal plots. This effect is seen in downstream planes as well. The reasoning behind this phenomenon will be explained in the Plane 3 discussion as the effect is more clearly identified and analyzed on that plane.

The thermal measurements for Planes 1 and 2 were taken for more than one pitch length (from Y/Pi =1 to Y/Pi = -0.05). This was done because a bleed slot is located upstream of the cascade that affected the flow in the upper passage differently than in the lower passage. Therefore, the data above Y/Pi = 0.95 were affected by this feature. Hence, the effects in that region should be neglected and the results for the region between Y/Pi = 0 and Y/Pi = -0.05 (located in the lower passage) should instead be used to understand the coolant transport in this region.

**Plane 3 (X/Cax = 0.289)**

Due to limitations of the facility, the aerodynamic data on this plane are limited to the region below Y/Pi = 0.61. Figures 5.3.6 through 5.3.9 show the thermal and aerodynamic contours. Based on the results found in Alqefl et al. (2021) [37-39] and Nawathe et al. (2021) [40,41], the flow on this plane had counterclockwise (positive) vorticity. This was a result of the combustor coolant flows causing coolant migration from the suction surface to the pressure surface away from the endwall (toward the mid-span). The vortex generated due to this secondary flow was termed as the ‘impingement vortex.’ Although introduced in a different fashion, the combustor coolant is present for the current study as well. Hence, the counterclockwise vorticity was expected on this plane. This is seen, but a clear
center of the vorticity is not observed, unlike the cases in Alqefl et al. (2021) [37-39]. There are two possible explanations to this phenomenon. One explanation is that the center of the vortex lies above $Y/Pi = 0.61$, in the region where measurements were not recorded. The other explanation is that the combustor coolant MFRs for the current cases are lower than that of the previous cases, which delays the formation of a coherent impingement vortex until downstream of the axial location of Plane 3. As will be shown later, for current cases, Plane 4 indeed shows the impingement vortex, implying that the mechanisms behind the generation of positive vorticity on Plane 3 for the current cases are similar to those described in earlier studies.

In Nawathe et al. (2021) [40,41], it was suggested that the effusion coolant (part of the combustor coolant) flow established the characteristics of the impingement vortex. For that study, the effusion coolant flowrate was constant for all cases. The current studies change the mass flowrate of the injected coolant that is equivalent to the effusion coolant from the previous study. The vorticity plots (fig. 5.3.7) show that an increase in the combustor coolant MFR leads to an increase in the magnitude of the positive vorticity near the endwall (see $Y/Pi = 0.5$). The magnitudes of the secondary flow vectors pointing toward the pressure surface near the endwall also increase with coolant flowrate. It is known that the impingement vortex forms, in part, due to its resistance to passage turning. Hence, higher positive vorticity and secondary flow vector magnitudes can be thought of as indicators of more resistance to turning. As this resistance is a function of the momentum of combustor coolant, it is logical to see that higher MFR cases (which have higher momentum) show greater resistance to passage turning. Additionally, the positive vorticity has a longer spanwise presence along the pressure surface for the higher MFR cases, indicating that not only the strength but also the size of the impingement vortex increase with increasing coolant flowrate. The vorticity magnitude near the pressure surface also increases with increasing MFR values. This causes flattening of the streamlines near this region, which is expected to affect coolant transport. This observation is important because it shows that a
degree of control over one of the most dominant secondary flows can be achieved by correctly choosing the coolant MFR.

Figure 5.3.6 Plane 3: Absolute velocity contours
Figure 5.3.7 Plane 3: Vorticity contours with secondary flow vectors
Figure 5.3.8 Plane 3: Thermal contours
Figure 5.3.9 Plane 3: Thermal contours with secondary flow streamlines
When observing the thermal contours (fig. 5.3.8 and 5.3.9), one can see an increase of coolant spread near the pressure surface in the spanwise (Z) direction from about Z/Si = 0.28 for the C52 case to about Z/Si = 0.4 for the C66 case. In previous studies, this level of change in the spanwise coolant spread was not observed and hence, it is attributed to changes in the strength and size of the impingement vortex. Along the endwall, the vorticity magnitude increases with increasing MFR values, but it does not seem to have any significant influence on the coolant distribution. For 0.6 > Y/\Pi > 0.22, the coolant layer distribution looks similar for all cases. The only difference is that the higher MFR cases have higher coolant recovery temperatures. But this is only because these cases had more coolant injected, which causes a loss of its cooling potential more slowly than in the lower MFR cases. Therefore, the impingement vortex is expected to have more direct influence on pressure surface phantom cooling than on endwall cooling.

For previous cases, hints of endwall crossflow (from pressure surface to suction surface) were present, based on the negative vorticity observed along the endwall. Such vorticity is not present for any of the current cases, except for a small negative-vorticity region seen at the endwall-pressure-surface corner. This implies that coolant migration from suction surface to pressure surface does not occur, even along the endwall at Plane 3. This can also be seen in thermal contours where the coolant layer thickness along the endwall starts decreasing for Y/\Pi > 0.6 for all cases as some coolant at these locations is being carried toward the pressure surface due to the counterclockwise rotation of the impingement vortex. Fortunately, some coolant remains along the endwall over the whole pitch, implying that the endwall at all locations on this plane will be receiving some amount of coolant. This thinning of the coolant layer above Y/\Pi > 0.6 is most apparent in the lowest MFR case, becoming less noticeable with increasing coolant MFR. This is because the momentum of coolant with a higher MFR resists migration toward the pressure surface more successfully and partially retains its presence near the suction-surface-endwall corner. Although aerodynamic measurements near the suction surface are not available, that coolant migrates toward the pressure surface implying that the passage vortex (which migrates
coolant from pressure surface to suction surface) is not present on this plane for any of the cases. The passage vortex would have been an evolution of the pressure leg of the horseshoe vortex, but the latter was not seen in this study. Hence, the absence of a passage vortex is understandable and is in line with the results of the previous studies. The reason behind the absence of endwall crossflow will be explained in the discussion of the Plane 4 results, as it is easier to discuss it there.

Figure 5.3.2 shows the contours of the ratio of the local velocity to the average inlet velocity for each of the cases at Plane 2. If one were to look at any one pitch location (Y/Pi) for the C52 case, they would see that the value of this ratio near the endwall is lower than the value in the mainstream. The value near the endwall correlates with the coolant velocity while the value away from the endwall correlates with the freestream velocity. This suggests that the coolant-to-mainstream velocity ratio at the inlet plane is lower than unity for the C52 case. Similarly, it can be observed that the velocity ratio is close to one for the C56 case and greater than one for the C61 and C66 cases. For the impingement vortex to occur, the momentum (and hence, velocity) of the coolant flow must be more than the mainstream momentum, so that the coolant can offer more resistance to passage turning than the mainstream does, which results in a swirl that leads to the impingement vortex. So, for cases with velocity ratios less than unity, the passage vortex should be the most dominating secondary flow observed. However, the velocity contours on Plane 3 (fig. 5.3.6) do not show any difference in distribution between the C52 and C66 cases. The C52 case shows that, at that plane, the coolant velocity is higher than the mainstream velocity and will favor the formation of the impingement vortex. This acceleration of the coolant flow between Planes 2 and 3 is caused by contouring the endwall. If the endwall in this study were flat, the impingement vortex would likely not have formed for the C52 case. This shows that contouring of the endwall is essential for good cooling performance if the coolant flow rates are generating a velocity ratio having a value close to, but less than, one. It is expected that, for coolant flow rates lower than the values considered in this study, even acceleration due to endwall contouring will
not be enough to achieve a velocity ratio of greater than one in the passage, and a passage vortex will be observed in place of the impingement vortex. This hypothesis was confirmed by taking measurements for such cases. The discussion of these cases is presented in Chapter 6.

**Plane 4 (X/Cax = 0.612)**

Figures 5.3.10 through 5.3.13 show the aerodynamic and thermal contours for Plane 4. The velocity contours (fig. 5.3.10) show that the flow has gone through strong acceleration downstream of Plane 3. The velocities continue to be the highest near the suction surface and are among the highest anywhere in the passage. They appear to be unaffected by changes in coolant flowrate.

This is the most upstream plane location on which the impingement vortex can be seen clearly in all cases. The center of the vortex is visible for all cases in the streamline plots. The most important observation on this plane is regarding the location of the center of the impingement vortex. Its location is approximately (0.2,0.6) for the C52 case and moves to (0.22,0.64) for the C66 case. Also, the location of the center of the impingement vortex for the C66 case approximately coincides with the locations of the centers for all cases from the previous studies. This means that lower MFR values create an impingement vortex nearer to the endwall. This displacement can be explained by the coolant momentum. For the lower MFR cases, the coolant injected far upstream of the vane passage remains close to the combustor-turbine interface as it does not have enough momentum to penetrate the mainstream. Unlike previous studies, this study has no gap in the interface section and, hence, the coolant forms a thick layer on the combustor wall. When this layer enters the passage, the coolant flow generates the impingement vortex due to the mechanisms of formation of impingement vortex, but it does so nearer to the endwall. For the higher MFR cases, by the streamwise distance the coolant reaches the vane passage, it has moved significantly in the spanwise direction. Hence, when the passage mechanisms force the coolant to form a vortex, it is formed slightly away from the endwall. Also, as the effusion coolant MFR was not changed for the previous studies (main contributor to the formation
of impingement vortex), the location of the impingement vortex did not change for any of the cases in those studies.

The displacement of the impingement vortex center may seem trivial, but its effects on other secondary flow features are significant. The first effect relates to the extent of the positive vorticity in the spanwise direction along the pressure surface. On Plane 3, it is seen that this extent increases with increasing MFR.
Measurements on Plane 4 suggest that this increase is also a function of the displacement of the region of influence of the impingement vortex. Thermal measurements show that the coolant travels further in the spanwise direction in cases with higher MFR values, confirming the above discussion.

The second effect relates to endwall crossflow. In the study by Nawathe et al. (2021) [40,41], a region of negative vorticity indicating crossflow along the endwall was seen clearly for all cases. Also, it was shown in previous studies that a higher strength of the impingement vortex meant reduced endwall crossflow. The current studies have lower coolant MFR values that generate lower magnitudes of positive vorticity implying a weaker impingement vortex. Hence, endwall crossflow in these cases should be more pronounced. Instead, none of the cases show negative vorticity along the endwall except near the endwall-pressure-surface corner implying that the impingement vortex eliminated coolant migration to suction surface. This seems contradictory if the discussion is limited to the strength of the vortex. In the current study, the location of the center of the vortex is closer to the endwall. This means that the effect of the impingement vortex on coolant very close to the endwall is more significant than in the previous studies. In effect, the impingement vortex resists the endwall crossflow. This resistance depends on the coolant flowrate. There is no clear negative vorticity region for the C52 case, only hints of it for the C56 case while the C61 and C66 cases show a considerable region of strong negative vorticity near pressure-surface-endwall corner. This is because of the impingement vortex moving away from both the endwall and the pressure surface with increasing coolant flowrate. As the vortex moves away from these surfaces, the negative vorticity appears at locations where the influence of the impingement vortex is diminished. Beyond these locations, the impingement vortex overwhelms the negative vorticity generated due to crossflow, resulting in a flow with positive vorticity.
The negative vorticity regions for any of these cases do not reduce the cooling effectiveness near the endwall, since their area of influence is small. Although the cases considered in this study have fixed injection angles, the results on Plane 4 suggest that coolant effectiveness distributions along the endwall and pressure surface can be controlled by changing the flowrate and injection characteristics of
the combustor coolant. Compared to the previous studies, the coolant thickness along the endwall is smaller in the current study, owing to an overall lower injection flowrate. However, values of the recovery temperatures near and along the endwall and the pressure surface are comparable with those of previous studies. These previous studies had a significant amount of coolant away from the surface, which did not aid in cooling throughout the passage but only helped in keeping the

Figure 5.3.12 Plane 4: Thermal contours
coolant layer along the vane passage surfaces intact. In the current cases, due to the proximity of the impingement vortex to the endwall and pressure surface, the need for a thick coolant layer is eliminated. This shows that the current combustor-turbine interface can provide the same level of cooling with a lower coolant flowrate, compared to the engine-representative combustor-turbine interface used in previous studies.

Near the suction-surface-endwall corner of Plane 4, the thermal contours for Cases C56, C61, and C66 (fig. 5.3.12) show a tendency for the coolant to run along the suction surface toward the midspan. This tendency increases when the coolant flowrate is increased. Such tendency was also observed for Plane 3 plots (see fig. 5.3.8). On the vorticity plots for Plane 4 (fig. 5.3.11), hints of negative vorticity can be seen near this region as well (velocity measurements very close to the suction surface are not possible with a five-hole probe). This implies that the flow has some clockwise rotation. This region falls outside of the influence of the impingement vortex. In Nawathe et al. (2021) [40,41], the negative vorticity was a result of the endwall crossflow turning in the clockwise direction near the suction surface after it was out of the region of influence of the impingement vortex. This phenomenon was termed ‘crossflow-induced vortex.’ However, for the current study, the endwall crossflow does not span the whole pitch (Y) and hence, there is no significant coolant migration toward the suction surface that can result in a crossflow-induced vortex. An observation of the streamline at the endwall near Y/Pi = 0.84 shows that the flow is impinging upon the wall (especially Cases C52 and C56). For Y/Pi > 0.84, the positive vorticity reduces in magnitude and ultimately becomes negative close to the suction surface. So, it is reasonable to assume that the flow impinges on the endwall and moves in two directions: one toward the pressure surface and the other toward the suction surface. While the pressure surface side merges with the impingement vortex, the suction surface side hits the suction surface and then moves along the suction surface toward the midspan, generating a negative vorticity region in the suction-surface-endwall corner. As previous cases had the crossflow-induced vortex (which has the same direction of rotation and general location as the negative vorticity being discussed), the effects
of this endwall impinging flow could not be observed in isolation from the crossflow-induced vortex. This negative vorticity region has two chief effects. The first is that it can provide resistance to the impingement vortex that can reduce the region of influence of the latter. The second is that it will help phantom cooling of the suction surface by increasing coolant coverage in the spanwise direction. The

Figure 5.3.13 Plane 4: Thermal contours with secondary flow streamlines
measurements on this plane show that there are multiple sources behind the formation of each of the secondary flow structures.

**Plane 6 (Throat Plane; starts at X/Cax = 0.612 at the Suction Surface)**

Plane 6 is located approximately at the throat of the passage. Figures 5.3.14 and 5.3.17 show vorticity and thermal contours of this plane. As this plane is close to Plane 4 but is much smaller in area, the effects seen on Plane 4 are expected to be amplified. Like Plane 4, the velocities on this plane have the same trends as for all cases of various MFR values. It appears that the velocities near the suction surface are decreasing slightly with increasing coolant flowrate. However, it should be noted that the plots represent velocities that are non-dimensionalized with the average passage inlet velocity for that case. As this quantity in the denominator varies from case to case, small differences in the velocity contours among different cases are expected.

All cases show a region dominated with positive vorticity (fig. 5.3.15) implying that the effects of the impingement vortex are present on this plane. The center of the vortex, which is seen on Plane 4, is visible clearly on this plane as well (fig. 5.3.16). The coolant spread along the pressure surface due to the impingement vortex is still present, which means that the pressure surface phantom cooling performance will largely remain similar to that at upstream locations. The positive vorticity region suggests that, except very near the suction surface (where measurements are lacking due to five-hole probe limitations), the impingement vortex spans the whole pitch (Y) proving to still be a dominant secondary flow feature.

Endwall crossflow near the pressure-surface-endwall corner, first seen on Plane 4, is also present on this plane. On Plane 4, the vorticity magnitude and size of this region increase with increasing coolant flowrate. On this plane, Cases C61 and C66 have a larger region with negative vorticity compared to C52 and C56. The shift in the centers of the impingement vortex with increasing coolant flowrate is in line with the centers observed for each case on Plane 4 and hence, the reasoning given for Plane 4 results as an increase in the endwall crossflow with
increasing coolant MFR will also apply for Plane 6. The increase in the crossflow intensity at Plane 6 compared to that of Plane 4 implies that in the streamwise direction, the impingement vortex is losing its coherency. Also, as on Plane 4, the change of vorticity near the endwall-pressure surface does not affect coolant transport on Plane 6, as seen in the thermal plots.

A negative vorticity region was observed on Plane 4 near the suction-surface-endwall corner. The vorticity plots of Plane 6 do not show a clear negative vorticity near this region. But the vectors near the endwall for $Y/Pi > 1$ are pointing toward the suction surface. Additionally, the thermal plots show the same tendency of coolant running along suction surface toward the midspan, as shown by the vectors and as seen on Plane 4. It will be shown that the exit plane (Plane 5) also shows this negative vorticity region, implying that this vorticity is present also on Plane 6, albeit only confined to a region very close to the suction surface where aerodynamic measurements cannot be taken. Finally, a low-effectiveness spot close to the pressure-surface-endwall corner can be seen only for the C66 case. The vorticity plots show no unusual behavior in this region. Due to the small size of the spot, it is believed that this could be an upstream effect of the wake of the trailing edge. It is possible that the small spots appear for all cases and the probe (due to the finite distances between successive measurements) was able to capture it only for one of the cases. The effectiveness values are lower by about 0.1 for this region than for the neighboring regions, but they are still high enough to prevent thermal damage to the endwall. Also, as the wake region of the vanes may not be correctly simulated by the test facility, such a low-effectiveness spot may not be present in the engine.
Figure 5.3.14 Plane 6: Absolute velocity contours

<table>
<thead>
<tr>
<th>|V|/U_{\text{Inlet}}</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.8</td>
</tr>
<tr>
<td>4.2</td>
</tr>
<tr>
<td>3.6</td>
</tr>
<tr>
<td>3</td>
</tr>
<tr>
<td>2.4</td>
</tr>
<tr>
<td>1.8</td>
</tr>
<tr>
<td>1.2</td>
</tr>
</tbody>
</table>
Figure 5.3.15 Plane 6: Vorticity contours with secondary flow vectors
Figure 5.3.16 Plane 6: Thermal contours
Figure 5.3.17 Plane 6: Thermal contours with secondary flow streamlines
Plane 5 (X/Cax = 1.02, Passage Exit Plane)

Figures 5.3.18 through 5.3.21 present the aerodynamic and thermal contours for the exit plane. As expected, the velocity profile is mostly flat for all cases (fig. 5.3.18). The negative-vorticity region near the suction-surface-endwall corner causes some mixing and locally reduces the absolute velocity. The flow near Y/Pi = 1 is in the wake of the vane. The flows along the pressure surface and suction surface mix in this wake region and cause a decrease in the velocity. The data in this region may not reflect the actual flow in the engine as the region downstream of the vanes in the facility does not accurately simulate the engine. The results in this region are presented only for completeness.

An observation of the streamlines in the region 1.4 > Y/Pi > 1.2 (fig. 5.3.21), shows that they appear to turn more sharply toward the pressure surface with increasing coolant flowrate. This is because the higher MFR cases generate a stronger impingement vortex and, hence, the mean vorticity of the flow is more for higher MFR values. The stronger vorticity in higher MFR cases also leads to a larger spanwise distance of coolant coverage along the pressure surface, as seen in the thermal contours (fig. 5.3.20).

When compared with the previous studies, the vorticity magnitudes on Plane 5 are lower. It is attributed to lower coolant flowrates in the current cases. Lower coolant flowrates imply lower momentum of the coolant which will result in less resistance to mixing with the mainstream. This would lead to a faster decay of the impingement vortex. The other reason behind why the impingement vortex is weaker at Plane 5 for the current cases is related to the coolant injection location. In previous studies, there were two rows of holes injecting endwall film coolant immediately upstream of the vane passage. These provided a continuous energizing of the impingement vortex, allowing it to sustain itself for longer distances in the passage. In the current study, there is only one row of coolant holes immediately upstream of the endwall. This means that the energy imparted to the impingement vortex is lower in the current study, which leads to a faster decay of the vortex.
Figure 5.3.18 Plane 5: Absolute velocity contours
Figure 5.3.19 Plane 5: Vorticity contours with secondary flow vectors
Figure 5.3.20 Plane 5: Thermal contours
Figure 5.3.21 Plane 5: Thermal contours with secondary flow streamlines
Similar to the behavior on the upstream planes, Plane 5 shows endwall crossflow for all cases. The crossflow has higher vorticity closer to the pressure surface. The vorticity becomes more intense with increasing MFR values. This increase has already been explained (see Plane 4 discussion) to be a result of the impingement vortex moving away from the endwall when coolant flowrates are higher, allowing endwall crossflow to increase in intensity. The endwall crossflow is less intense and has reduced coverage in the pitch direction than in previous studies. This means that less coolant will migrate toward the suction surface. The coolant concentration contours show that the thickness of the coolant layer along the endwall is lower in the current cases compared to previous cases. However, immediately along the endwall, the values of recovery temperature are pitchwise uniform and comparable to previous studies. Hence, endwall cooling in the current cases is not negatively affected by suppression of endwall crossflow.

In the upper half of the plane, endwall crossflow intensity decreases while the effects of negative vorticity generated due to flow impinging on the endwall (see Plane 4 for discussion) are becoming apparent. This negative vorticity region is much smaller on the upstream planes. As the effects of the impingement vortex have decreased significantly at the exit plane, the size of the negative vorticity region increases. The vorticity magnitude in the suction-surface-endwall corner seems comparable for all cases, implying that the coolant flow rates do not have strong effects on this feature. This negative vorticity leads to coolant spreading along the suction surface in the midspan direction for all cases, as seen in the thermal contours (fig. 5.3.20). This is expected to increase coolant coverage along the suction surface, as compared to previous studies, which will be discussed in a following section.

5.4. Vane Passage Surface Effectiveness Measurements

Endwall Adiabatic Cooling Effectiveness

Figure 5.4.1 shows the endwall cooling effectiveness contours for all cases. Near the leading edge, cooling effectiveness values increase slightly with increasing coolant MFR. The coolant distribution is pitchwise uniform, except near
the suction surface where slightly elevated values of effectiveness are recorded. Near X/Cax = 0.3 (near Plane 3), a drop in effectiveness values can be seen near the suction surface, compared to near the pressure surface. This drop is most visible for the C52 case and starts to disappear with increasing coolant flowrates. Case C66 shows no drop. Near Plane 3, turning of the suction surface leads to a decrease in velocity of coolant flow along the endwall. This will cause a reduction in momentum of the coolant, which would make the coolant more susceptible to the effects of the impingement vortex. A look at the Plane 3 thermal flowfield measurements (fig. 5.3.8) shows coolant thinning along the endwall near the suction surface. This thinning occurs because the positive vorticity of the impingement vortex transports coolant from the suction surface region to the pressure surface region. This transport is the cause of reduced coolant coverage on the endwall near the suction surface. As the coolant thinning effect decreases with increasing coolant MFR, the drop in effectiveness on the endwall also starts to diminish with increasing MFR. In previous studies, the region near Plane 3 showed higher coolant effectiveness near the suction surface compared to that near the pressure surface (fig. 5.4.2). This was a result of coolant migration toward the suction surface occurring due to endwall crossflow. As the current cases do not show endwall crossflow in this part of the passage, such coolant migration is not observed. Consequently, endwall cooling performance near the pressure surface increases considerably in the current cases, which is shown in the literature to be a difficult region to cool.

However, downstream of Plane 3, this drop in effectiveness disappears for all cases. That is because downstream of Plane 3 a negative vorticity region is observed near the suction-surface-endwall corner (fig. 5.3.11). This means that some coolant that is much closer to the suction surface than the pressure surface will be affected due to the negative vorticity near the suction surface. This would lead to more coolant near the suction surface for locations downstream of X/Cax = 0.3 compared to locations upstream. Hence, the negative vorticity region is helpful toward generating a pitchwise-uniform distribution of coolant effectiveness along the endwall. Such uniformity is observed for all the cases.
Figure 5.4.1 Endwall adiabatic cooling effectiveness of current study
Near the trailing edge, the cooling performance increases marginally with increasing coolant MFR. The effectiveness values are about 0.6 for the C52 case and are about 0.65 for the C66 case. These values are well above the values that are observed in conventional cascades that have the passage vortex secondary flow system and not the impingement vortex. When Cases C56 and C61 are compared with cases from the previous studies with similar coolant MFR (fig. 5.4.2), the current cases show higher effectiveness values near the trailing edge, especially in the suction-surface-endwall corner. There are two explanations for this improved performance. Firstly, in some of the cases in the previous studies,
momentum of the coolant was high enough to penetrate the mainstream. This caused mixing of coolant with the mainstream that led to reduced performance along the endwall. The exit plane flowfield measurements in the current study do not show mainstream penetration and, thus, there is no significant drop in cooling performance near the trailing edge for this study. Secondly, the endwall crossflow was more severe in the previous studies. This crossflow generated a vortex near the suction surface that migrated coolant away from the endwall and along the suction surface in the midspan direction. Such a phenomenon occurs in the current study as well, but its intensity is much lower compared to the intensity in the previous cases. In effect, a performance drop near the suction-surface-endwall corner does not occur in the current cases, leading to a pitchwise-uniform distribution, even at the trailing edge.

When two equivalent cases, Case C56 (from the current study, fig. 5.4.1) and L09F00 (from the previous study, fig. 5.4.2) are compared, the current study showed better performance. This is because the number of rows immediately upstream of the vane passage were different for the two cases. Case C56 had one row of coolant holes blowing immediately upstream of the endwall while Case L09F00 had no such rows and no endwall film coolant injection. Injection through the most downstream row helps in preserving the coolant film along the endwall for a longer distance, providing better performance for the present case. Similarly, when Cases C66 and L09F10 are compared, the case from the previous study shows better results in the upstream part of the passage because that case has two rows blowing coolant, compared to the single row of the present case, Case C66. However, the performance in the downstream half is not different between these cases and the high performance seen in Case L09F10 at the upstream part of the passage may not be required in many engines. These observations show that having at least one row of coolant blowing very close to the vane passage can provide significant improvement in endwall cooling.

For Cases C61 and C66, low-effectiveness spots are observed in the pressure-surface-endwall corner near the trailing edge. For both C61 and C66 measurements, only one measurement location each recorded low values of
effectiveness (by about 0.15 compared to the neighboring locations). On Plane 6, three measurement locations in the same region recorded lower effectiveness values. It seems that this low-effectiveness spot changes its location with increases in coolant flowrate. This was seen at Plane 6 for only the C66 case and not seen on Plane 6 measurements for other cases. Also, aerodynamic measurements at Plane 6 for the C66 case do not show unexpected secondary flow activity in this region. Strong endwall crossflow is present in this region for all cases. Although a clear explanation behind the formation of these spots cannot be given, the above observations suggest that this is very localized behavior. The values of the effectiveness, although lower than in neighboring regions, still are higher than 0.5. Hence, thermal damage to the endwall at these locations is not expected in the engine.

Throughout the passage, the rate of decay of the coolant effectiveness is low. In other words, the endwall cooling performance is sustained for the whole passage. As the effects of the impingement vortex are present for most of the passage, severe coolant migration toward the suction surface due to the passage vortex seen in conventional cascades, is not present for the current study. Additionally, as the impingement vortex remains close to the endwall, the endwall crossflow, although present, never becomes more dominating than the impingement vortex. Hence, the coolant migration that occurs due to endwall crossflow is minimal and proves to be helpful in providing pitchwise-uniform performance, as discussed earlier. When compared to previous studies, for same coolant flow rates, the rate of decay of coolant performance was higher for previous studies. If Case L09F10 from Nawathe et al. (2021) [40,41] and the Case C56 from the current study are compared (see fig. 5.4.1 and 5.4.2), they show similar cooling performance in the downstream half of the passage. This happens because the cases in the previous study had either too much or too little coolant momentum. The cases with too low momentum resulted in coolant succumbing to the passage secondary flows and mixing with the mainstream, leaving the endwall without cooling. The cases with too high momentum made the coolant penetrate the mainstream, removing the coolant film along the endwall. For the current study,
cases with the coolant flowrates considered show that the coolant film remains attached to the endwall for longer axial distances. Hence, Case L09F10 (which had a total coolant MFR of 6.64%) and Case C56 (which had coolant MFR of 5.6%) showed comparable performance, despite the latter case having much lower coolant flowrate. This comparison shows that the current close-coupled interface coolant injection scheme is making more efficient use of the injected coolant than the engine-representative cooling scheme used in previous studies.

**Pressure Surface Adiabatic Cooling Effectiveness**

The combustor coolant was found to cool the pressure surface of the vanes considerably. This unintentional cooling is called 'phantom cooling.' Figure 5.4.3 shows adiabatic effectiveness contours of the pressure surface for all cases. The cooling effectiveness values are increasing with increasing coolant MFR. This is expected as more coolant is available near the pressure surface for higher MFR cases as the coolant streams are directly impinging on the pressure surface. Additionally, the spanwise (Z) spread of the coolant away from the endwall also increases with increasing coolant MFR. This is a result of the size and strength of the impingement vortex. The flowfield results show that the impingement vortex grows stronger and larger in size as the coolant flowrate is increased. Its location also moves away from the endwall (i.e., in the positive Z direction). This leads to an increase in the spanwise coolant spread on the pressure surface. It is worth noting that the previous studies showed an impingement vortex of similar size and strength for all cases and, consequently, the pressure surface contours for all cases had equal spanwise coolant spread. Hence, changing coolant injection rate for the newer combustor-turbine interface can lead to more benefits to pressure surface phantom cooling as well.

The trends of cooling effectiveness near the endwall upstream of X/Cax = 0.15 show that the C52 case behaves differently than other cases. This case shows a reduction in the coolant layer thickness in the spanwise direction compared to the thickness at the leading edge. On the other hand, the other cases show a monotonic increase in the coolant thickness from leading edge to X/Cax = 0.15. Downstream of this location, all cases start showing similar trends. In the flowfield
results, Plane 2 velocity contours showed that the coolant velocity was lower than the mainstream velocity for the C52 case, comparable for the C56 case and higher for the C61 and C66 cases. Case C52 has a coolant-to-mainstream velocity ratio of less than 1, which is the reason behind the different trend of cooling performance on the pressure surface. For the C52 case, the fluid away from the endwall will travel toward the endwall as the coolant nearest to the endwall has lower stagnation pressure (a mechanism that is similar to the one that forms the pressure leg of the horseshoe vortex in conventional cascades). This can lead to the mainstream flowing toward the endwall and mixing with the coolant to reduce
cooling effectiveness. As the flow travels downstream, the contour of the endwall provides enough acceleration to the coolant near the endwall for its velocity to become more than that of the mainstream. Hence, the phenomenon of coolant impinging on the pressure surface and rushing toward endwall stops. Instead, the coolant will start traveling away from the endwall in the midspan direction (which is one of the mechanisms of formation of the impingement vortex). The positive vorticity seen on Plane 3 near this location (see fig. 5.3.7) verifies that an
impingement vortex starts to form for the C52 case. Similar to the flowfield measurements, the pressure surface effectiveness measurements also show that it is important to achieve a coolant-to-mainstream velocity ratio higher than 1 as far upstream in the passage as possible.

Around $X/Cax = 0.2$, all cases show an abrupt drop in performance. This is most visible in Cases C56 and C66. The closest flowfield measurement is at $X/Cax = 0.289$, which does not show unusual behavior. As there were many surface effectiveness measurements taken at this location for all cases, it is believed to not be an error in recording. Most probably, while spreading the coolant along the pressure surface, the impingement vortex could be lifting the coolant off the pressure surface, causing a small drop in effectiveness. The reason why this happens at only this location could be that the geometry of the vane pressure surface at only that location is favorable for such lifting. Irrespective of the reason behind this effectiveness drop, it can be concluded that it will always occur at the same axial location and that it will be highly localized.

The coolant layer thickness increases in the spanwise direction as the flow travels through the passage until $X/Cax = 0.75$. Downstream of this location, the coolant effectiveness drops, and the coolant layer thickness no longer increases. This is observed for all cases at the same location, but with varying degrees. There are multiple causes with this phenomenon. Based upon the flowfield measurements, the impingement vortex begins to dissipate after Plane 4 ($X/Cax = 0.612$). This was happening as the direct impinging of the coolant on the pressure surface had decreased significantly at this point in the passage. This impinging of flow on the pressure surface energized the impingement vortex. Without it, the impingement vortex strength decayed downstream. Because of this, less coolant was being washed along the pressure surface, leading to a decrease in the coolant effectiveness. As the impingement vortex was strongest in the highest MFR case, the coolant effectiveness decrease on the pressure surface is least significant for that case. Additionally, downstream of $X/Cax = 0.6$, the contour of the endwall becomes concave, which would decelerate the coolant flow near the endwall. This deceleration can create coolant mixing due to formation of small a recirculation
zone (which transports flow from midspan toward the endwall) reducing coolant effectiveness. Finally, the most upstream detection of endwall crossflow was at Plane 4 ($X/C_{ax} = 0.612$). This crossflow is expected to drive coolant along the pressure surface toward the endwall and then along the endwall and toward the suction surface. Hence, the crossflow can cause a reduction in the cooling performance on the pressure surface in the region downstream of Plane 4. At the trailing edge, for all cases, effectiveness values of about 0.6 and 0.5 are observed near the endwall and at 30% of the span away from the endwall, respectively. These values are comparable with the values recorded at the exit of the passage for endwall contours.

When the pressure surface contours of the current study are compared to those of the previous studies (fig. 5.4.4), it is found that the former gives better cooling effectiveness over a longer spanwise distance. As the impingement vortex is closer to the pressure surface in current studies, its effect on coolant spreading along the pressure surface is more pronounced. As the endwall crossflow is delayed until downstream of $X/C_{ax} = 0.612$, there is less tendency for coolant to travel toward the endwall, which results in better retention of coolant near the pressure surface. Finally, similar to the discussion on endwall cooling, the coolant flowrates considered in this study favor sustained coolant film attachment compared to the cases from the previous studies. Overall, the combustor cooling injection scheme employed in current studies is also helpful in increasing phantom cooling of the pressure surface.

**Suction Surface Adiabatic Cooling Effectiveness**

Effectiveness contours for suction surface phantom cooling are shown in fig. 5.4.5. As the impingement vortex was the dominant secondary flow for most of the vane passage, no significant coolant migration toward suction surface was observed. Hence, in general, the coolant coverage along the suction surface is much smaller than the coverage along pressure surface. Up to $X/C_{ax} = 0.2$ and due to the geometry of the vanes, coolant directly impinges on the suction surface and hence, high effectiveness values are seen on the suction surface near the endwall. As the coolant MFR increases, the cooling performance near the leading-
edge region also increases as more coolant is impinging on the suction surface. Due to the proximity of the high-effectiveness coolant to the endwall, it is believed that this coolant was the coolant injected from the most downstream row of holes. Also, the cases from previous studies that had the same total coolant flow rate as those of Cases C56 and C61 from the current study, did not have any coolant blowing from the most downstream row of holes (these were cases that had an endwall film coolant MFR of 0%) and did not show a high-effectiveness coolant film on the suction surface near the endwall.

Figure 5.4.5 Suction surface adiabatic cooling effectiveness of current study
Beyond \( X/Cax = 0.25 \), the thickness of the coolant layer starts to decrease. This is around the location where an effectiveness drop on the endwall near the suction surface was observed. Hence, the reasons for both effects are the same. The impingement vortex effects start to become significant and draw coolant away from the suction surface. This leads to a decrease in the coolant layer thickness along the suction surface for all cases. However, downstream of \( X/Cax = 0.6 \), the coolant layer thickness starts to increase in the spanwise direction and keeps increasing up to the trailing edge. Flowfield measurements in this region of the figure show...
passage showed the presence of endwall crossflow and the negative vorticity region near the suction-surface-endwall corner. Additionally, the impingement vortex started to decay in this region. A combination of these two effects is that more coolant started to migrate toward the suction surface and hence, a coolant coverage increase is seen in the suction surface effectiveness plots. The increase in the thickness is marginally larger for cases with higher coolant MFR, owing to a slightly stronger negative vorticity region in the suction-surface-endwall corner.

When compared to the previous studies (fig. 5.4.6), the current studies show higher cooling performance on the suction surface for all cases. This can be explained by multiple phenomena. Like the previous studies, the current study shows a negative vorticity region near the suction-surface-endwall corner. However, unlike the previous studies, the current study does not show the presence of the suction side leg of the horseshoe vortex at X/Cax = 0.612. This horseshoe vortex was resisting the negative vorticity region observed in the previous studies. As this resistance is not present in the current study, more coolant is washing up along the suction surface. The region of influence of the negative vorticity in the endwall-suction-surface corner is similar for both the previous and the newer cases. Hence, the cooling effectiveness is higher for the current studies compared to previous studies, but the coolant coverage is similar for all of them.

5.5. Conclusions

This chapter discussed the performance of the coolant injection scheme of a close-coupled, combustor-turbine interface. Four combustor coolant mass flowrates were considered to understand how coolant momentum affects the coolant transport. Secondary flow velocity vectors and streamlines, recovery temperature coefficients, and surface effectiveness values at various axial locations were presented to completely characterize the vane passage flow. The major conclusions are summarized in this section.

The engine-representative combustor-turbine interface used in the previous study (Nawathe et al. (2021) [40,41]) had a cavity between the combustor and the
turbine sections, which generated a pitchwise (Y) non-uniformity in the injected coolant. A lack of this cavity in the close-coupled interface prevented this non-uniformity, leading to good coverage on the endwall along the whole pitch.

The impingement vortex was the most-dominant secondary flow feature for all cases of this study. This vortex has been observed in the previous study as well, which confirms that it is not dependent on the geometry of the combustor-turbine interface but rather dependent on whether the combustor coolant injection is present in the test facility. The size and intensity of the impingement vortex increased as the combustor coolant flowrate is increased. In the previous study, the endwall coolant flowrate was varied and such effects were not seen. Therefore, the current study confirms that combustor coolant plays a major role in the formation of the impingement vortex. Increasing the combustor coolant flowrate also moved the impingement vortex center away from the endwall-pressure-surface corner. This happens due to the tendency of higher momentum coolant to be more successful in penetrating the mainstream and, therefore, to move away from the endwall. For lower flowrates, the impingement vortex was sufficiently close to the endwall to eliminate the endwall crossflow completely in the upstream part of the passage. For cases in this study, this did not affect the coolant concentration along the endwall, but it could become important for cases with lower coolant momenta.

For all cases, a pitchwise-uniform, high-effectiveness coolant layer was present along the endwall throughout the passage. This layer was thinner than the one observed in the previous studies as the coolant flowrates in that study were higher. Still, the recovery temperature coefficients next to the endwall were similar for both cases. Due to the secondary flow patterns in the current study, the coolant layer was able to persist throughout the endwall even while being thinner than the previous study. This shows that the close-coupled interface requires lower coolant flowrates to provide similar levels of cooling as the engine-representative interface.

Based on the endwall effectiveness contours from previous and current studies, it is found that having at least one row of coolant holes immediately upstream of the vane passage considerably increases the cooling performance of the endwall
throughout the passage. The persistence of the coolant layer on the endwall and size and strength of the impingement vortex helped in providing better endwall cooling performance for the current study, compared to previous studies. For the current study, the proximity of the impingement vortex to the endwall helped in retaining more coolant near the pressure surface, which led to an increase in the spanwise direction and an increase in the pressure surface phantom cooling effectiveness. Higher coolant flowrates generated larger impingement vortices and increased the coolant spread along the pressure surface in the spanwise direction. Suction surface phantom cooling performance was much lower than that of the pressure surface performance due to a lack of coolant migration toward the suction surface. The suction leg of the horseshoe vortex was absent in the current study, which helped in marginally increasing the cooling performance, compared to that of previous studies. Substantial changes to the coolant injection scheme will be required if significant increases in the suction surface phantom cooling performance are desired.

A summary of the cooling performance of all cases on all surfaces is provided in the isometric view of the vane passage surfaces shown in fig. 5.5.1. Case C56 shows pitchwise-uniform endwall cooling that is sustained throughout most of the passage. Cases C61 and C66 show better cooling in the upstream half of the passage than in Case C56, but the downstream halves of all these cases have similar performance. Therefore, the requirement of additional coolant flowrate for Cases C61 and C66 is not justified. The phantom cooling effectiveness values on the pressure surface for Cases C56 through C66 are similar, although the spanwise coverage of the coolant away from the contoured endwall increases with increasing coolant flowrate. The phantom cooling performance on the suction surface is low irrespective of the coolant flowrate. Based on these observations, Case C56, which corresponds to a total coolant MFR of 5.6%, is found to have the most effective balance between the amount of coolant injected and the cooling performance on the vane passage surfaces. In the engine-representative, combustor-turbine interface study (Nawathe et al. (2021) [41]), the case with the total coolant MFR of 6.6% was found to be most effective. The decrease in the
requirement of coolant MFR from 6.6% to 5.6% shows that the close-coupled, combustor-turbine interface provides a more efficient coolant injection scheme than the engine-representative combustor-turbine interface.

Figure 5.5.1 Overall vane passage adiabatic cooling effectiveness. Note that the vanes extend in the positive Z-direction up to 50% of the inlet span.

6.1. Motivation

After analyzing the performance of the close-coupled, combustor-turbine interface in the previous chapter, it was suspected that if the combustor coolant flowrate is reduced further than the lowest flowrate considered in that study, the secondary flow structures may change significantly. It is possible that for such low coolant flowrates, the flowfield can revert to the passage-vortex-dominated secondary flow system (see Fig. 1.1.1). Such transition from the ‘impingement vortex’ system (observed in the previous chapter) to the ‘passage vortex’ system (observed traditionally in vane passage flows and discussed in section 1.1) has not been documented in the literature and this chapter records and explains this phenomenon.

6.2. Experimental Procedure

To document the passage flowfield, aerodynamic measurements were taken in the same fashion as done in the previous chapter. The calculation of secondary flows using the five-hole probe measurements is explained in section 3.2. Three coolant-to-mainstream mass flowrate ratios were chosen for the study. Additionally, Case C56 from the previous chapter, which corresponded to a coolant MFR of 5.6%, was used in this study as it shows a strong impingement vortex secondary flow system. Finally, a case with no coolant injection (MFR = 0%) was considered as a reference case because this case was expected to show a strong passage vortex secondary flow system. The case nomenclature used in this chapter is the same as in the previous chapter. Table 6.2.1 lists the cases in this study:
Three axial locations (planes) were chosen to take the measurements. These locations are the same as Planes 3, 4, and 5 from the previous chapter. Figure 6.2.1 shows these planes and their axial locations, which are non-dimensionalized with the axial chord length (X/Cax). The exact measurement locations are shown in fig. 6.2.2, which are similar to those of the previous study, except for Plane 5. Unlike in the previous study, no measurements for Y/Pi < 1.1 are plotted in the results section for Plane 5. This was done for two reasons. First is because this
region is influenced heavily by the wake of the vane trailing edge. Some of the cases considered in this study had weak secondary flows and these flows were more influenced by the wake. This wake is not similar to the wake generated in the engine as the downstream components in the engine (i.e., the rotor stage) are not simulated in the test facility. Therefore, the measurements taken in this region will not be accurate representations of the engine flow. The second reason will be explained in the next section.

6.3. Results and Discussion

This section is divided into the three subsections. The first two subsections explain a couple of issues that were faced during recording of these measurements and the ways in which they were resolved. The final subsection discusses the contour plots generated from all measurements taken as part of this study.

6.3.1. Repeatability in Generation of Secondary Flows

When the data recording for this study began, a particular pattern of data collection was used. It was the same as the one used for previous studies in this wind tunnel facility. This was an efficient way of recording the data, but it resulted in taking measurements for any plane over the course of a few days. The resulting contours for those studies (which had higher coolant flowrates than the current study) did not show any inconsistency between data recorded for a single plane on different days. However, when this data-recording pattern was used for the
current study while taking Plane 4 Case C26 measurements, the resulting vorticity contour with secondary flow vectors (shown in fig. 6.3.1 (a)) showed inconsistency between data taken on one day (all locations with $Y/P_i > 0.7$) and the data taken on the day after (all locations with $Y/P_i < 0.7$). At first, this was thought to be an error in measurements or data processing. However, all these procedures were verified to be performed correctly. The secondary flow vectors have low magnitudes for the coolant flowrates considered in this study. It was found that these weak secondary flows were generated slightly differently every time the wind tunnel was started.

As measurements for Plane 4 were taken on different days, there was inconsistency between secondary flows measured below and above $Y/P_i = 0.7$. Additionally, to calculate secondary flow vectors in fig. 6.3.1, a single reference vector was chosen for each plane, which was at the mid-span, mid-pitch location. Note that this method is different than the method explained in chapter 3, where measurements recorded at each pitch ($Y$) location (any horizontal row in fig. 6.3.1) had a separate reference vector that was recorded at the mid-span of that same pitch location. This single reference method was used to identify how much shift is present between velocity vectors recorded on different days. This method led to a reference vector that was measured on one day to be used to calculate the secondary flow vectors that were measured on the day after. As suspected, this led to the disparity between the contours generated from data measured on different days as seen in fig. 6.3.1 (a).

To ensure that the shutting down and restarting of the wind tunnel was the sole reason for inconsistent data, Plane 4 measurements for Case C26 were recorded for two additional times (called iterations). Each of these iterations was recorded in a single day. Measurements in only the region of the plane where the inconsistency was recorded were repeated. These are shown in fig. 6.3.1 (b) and (c). They show that the inconsistency near $Y/P_i = 0.7$ had disappeared completely. Both the vorticity magnitudes and secondary flow vectors did not show abrupt changes. Additionally, although iterations 1 and 2 were recorded on different days,
the difference in the overall secondary flow structure is essentially similar. This shows that, although restarting the wind tunnel changes the generation of secondary flows, this change does not affect flow physics. Therefore, although different planes and different cases are recorded across a span of few weeks, the results drawn from them will still give a consistent overall picture of the flowfield, so long as measurements on any individual plane were recorded without shutting down and restarting the wind tunnel. The remainder of the measurements

Figure 6.3.1 Plane 4 Case C26 vorticity contours. These were recorded to verify the repeatability of generated secondary flows
presented in this chapter were recorded in a way to avoid such data inconsistency, as will now be discussed.

6.3.2. Effect of Changing the Reference Pressure of Measurements

When the measurements were taken in a way described in the prior section, most of the data looked accurate and consistent. However, when measurements were taken on the exit plane (Plane 5 in fig. 6.2.1), sudden shifts in the velocity...
vector magnitudes and directions were observed for $Y/\Pi < 1.34$. These shifts for Cases C26 and C35 are shown in fig. 6.3.2 as representative cases. The measurements were taken without restarting the tunnel and no issues were observed while recording and analyzing the data. The secondary flow vectors were calculated in the same way that was used to plot the data in section 6.3.1 to eliminate the effect of changing reference vectors for measurements in each pitch location. The only difference between the measurements taken above and below $Y/\Pi = 1.34$ was the reference pressure that was used for collecting the data. As discussed in section 3.2, this reference pressure is measured at specific locations on the vane surfaces. During measurements, the reference pressures are changed according to measurement locations to ensure that the pressure applied around the pressure transducers is within operating limit. When the pressure coefficients ($C_P$) are calculated, the reference pressure term is cancelled, making the measurements independent of reference pressure.

For Plane 5, the reference pressure was changed after data were collected at $Y/\Pi = 1.34$. Unexpectedly, this brought a drastic change in the secondary flow vectors. To confirm that this happens only due to a change of reference pressure, measurements were repeated for the region $1.25 < Y/\Pi < 1.4$ while keeping the reference pressure the same. This test was done for two different probe insertion angles ($54.5^\circ$ and $64.7^\circ$) to ensure that the probe insertion angle did not affect the secondary flow vectors. The resulting contours of this test are shown in fig. 6.3.3. They show that the vectors remain generally the same in both magnitude and direction if the reference pressure is not changed. The vorticity is slightly more for the left contour, but the thickness of high vorticity region is the same for both cases. The probe insertion angle does not seem to have affected the secondary flow formation.

The secondary flow vector at any location is calculated by subtracting the ideal streamline vector from the velocity vector measured at that location. For fig. 6.3.2, this ideal streamline vector is chosen to be the velocity vector measured at the mid-pitch, mid-span location (see location R1 in fig. 6.3.4). It is established that the change of reference pressure resulted in inconsistency of secondary flow vectors.
Therefore, it was suspected that the measurements taken at Plane 5 below \( Y/\Pi = 1.34 \) were not entirely wrong but the measured pressures in the five-hole probe were all shifted by some amount. Due to this shift, when the ideal streamline vector (measured above \( Y/\Pi = 1.34 \)) is subtracted from these ‘shifted’ vectors (measured below \( Y/\Pi = 1.34 \)), the resulting secondary flow vectors became inconsistent with the secondary flow vectors generated for region above \( Y/\Pi = 1.34 \). Therefore, for measurements below \( Y/\Pi = 1.34 \), a different ideal streamline was chosen to verify whether the measurements are only shifted. With this new reference (see location R2 in fig. 6.3.4), the secondary flows were recalculated for this region and are plotted in fig. 6.3.4 along with the rest of the measurements above \( Y/\Pi = 1.34 \). For both representative cases, the secondary flow vectors now appear to be consistent.

Figure 6.3.3 Rerecording a fraction of measurements of Case C26 for verifying effect of changing reference pressure. Two separate probe insertion angles were considered.
with each other confirming that changing reference pressure shifted the measurements. When the measurements are plotted for final presentation, all secondary flow vectors at any pitch location are calculated by a reference vector that was at the mid-span of that pitch location (as described in section 3.2). Therefore, as the reference vector changes for each pitch row, the secondary flow

Figure 6.3.4 Replotting of Plane 5 measurements for Cases C26 and C35 by using two different ideal streamline vectors

R1: Streamline Reference Location for Vectors above Y/Pi = 1.34
R2: Streamline Reference Location for Vectors below Y/Pi = 1.34
vectors will not appear to be ‘shifted’ as seen in fig. 6.3.2. Nonetheless, the discussion presented in this subsection is meant to provide an understanding of the effect of changing the reference pressure on the recorded data and to advise caution to future researchers on the possibility of this occurrence.

Another solution for this problem was to record the whole plane using a single reference pressure so that the data would look consistent. However, this would make the pressure difference experienced by the transducers’ diaphragms connected to the five-hole probe to go well beyond its recommended range. This can introduce large uncertainty in measured data and can possibly damage the transducers. In the future, a transducer with a larger range of acceptable pressure differences could be used instead, to avoid this issue.

The changing of reference pressure midway through recording data on a plane has not affected any measurements in the previous studies. Also, such reference pressure change was performed for Plane 4 measurements of this study as well, and that did not result in any abrupt changes in velocity vectors. Therefore, this phenomenon seems to appear at the exit of the passage for a very specific range of coolant flowrates. Unfortunately, no explanation about why this event happens was found.

6.3.3. Discussion of Secondary Flowfield Measurements

For each plane, plots for all cases are presented together. For each case and each plane, three plots are presented: (a) Absolute velocity contours, (b) Vorticity contours with secondary flow vectors, and (c) Vorticity contours with secondary flow streamlines. The vectors provide the readers a sense of where the strongest and weakest secondary flows are located on any plane and how their magnitudes and directions change with coolant flowrate. The streamlines help in identifying various secondary flow structures on each plane.

Unlike Chapter 5 results, the velocity contours presented in this chapter are not non-dimensionalized using the passage inlet mean velocity of their corresponding cases. This was done because measurements at the passage inlet plane (Plane 2) were not recorded for this study as it was known that they will not be relevant to
the objectives of this study. As the passage inlet mean velocity changes with the injected coolant flowrate, values from the Chapter 5 study could not be used. Therefore, the velocity contours are plotted with their absolute magnitudes. The extremes of the contour legends for each plane approximately correspond to the non-dimensionalized velocity magnitudes of the contours presented in the previous study (see section 5.3). This means that the colors of the contours of the velocity plots at corresponding planes in Chapters 5 and 6 imply similar values of absolute velocities.

**Plane 3 (X/Cax = 0.289)**

The absolute velocity contours for Plane 3 are shown in fig. 6.3.5. The absolute velocities near the endwall are affected by coolant injection. For Cases C46 and C56, at any pitch (Y/\pi) location, the velocity near the endwall is higher than that away from the endwall. This shows that the coolant flow has higher momentum than the mainstream for these cases at Plane 3. For Case C35, the coolant and

Figure 6.3.5 Plane 3 velocity contours
mainstream velocities are almost equal, as shown by the lack of change in the colors of the contours at any pitch location when moving from endwall to midspan. The plot for Case C26 shows that the coolant velocity is lower than the mainstream. Case C00 does not have any coolant injection and hence, shows a rather flat velocity contour in the spanwise (Z/Si) direction. Note that the five-hole probe cannot take measurements close to the endwall and hence, the lower velocities in the boundary layer are not seen on any of these plots. Generally, these plots show that the coolant stays close to the endwall. The absolute velocities are lowest near the pressure surface and increase gradually as one moves toward the suction surface. This is expected as the flow tends to accelerate more near the suction surface.

Figure 6.3.6 Plane 3 vorticity contours with secondary flow vectors and streamlines: Cases C00 and C26
Figures 6.3.6 through 6.3.8 show the vorticity contours with secondary flow vectors and streamline traces for Plane 3. The behaviors of the secondary flow vectors away from the endwall are similar for all cases. This includes both the directions and magnitudes of the vectors. The noticeable differences between the cases are seen in the near-endwall region. The vectors in this region become more directed toward the pressure surface with increasing coolant flowrate. For Cases C00 through C35, the vectors near the endwall point toward the suction surface. But they start pointing toward the pressure surface for Cases C46 and C56. This change happens because coolant with higher momentum resists the passage turning (passage turns in the figure upward direction in the contour) more.
successfully and hence, the vectors tend to be flatter for the C56 case compared to the C26 case.

For contours of Cases C00 and C26, the endwall crossflow is clearly visible by way of negative (clockwise) vorticity with a concentrated intensity in the region $0.4 < Y/Pi < 0.5$. This crossflow is induced due to the pressure gradient between the pressure surface (high pressure) and the suction surface (low pressure). Usually, a passage vortex is also seen near the pressure surface (negative vorticity) for such flows. But, because the Reynolds number (~370,000) and the inlet turbulence intensity (~10%) are very high, secondary flow vortices are known to dissipate (Chung and Simon 1993 [81]), leading to more non-coherent structures than the

Figure 6.3.8 Plane 3 vorticity contours with secondary flow vectors and streamlines: Cases C56
ones seen in low Reynolds number studies (Wang et al. (1997) [5]). Although a coherent passage vortex is absent on this plane, the streamlines tend to move clockwise (especially for Case C26), which is a direct effect of the passage vortex. Additionally, the effects of the endwall crossflow and the passage vortex tend be similar and in the vicinity of each other. Hence, it is not always possible to observe these two effects independently of each other at all locations in the passage. In general, the secondary flow systems for Cases C00 and C26 resemble a typical ‘passage-vortex system’ seen in the literature.

The endwall crossflow is expected to decrease with increasing coolant flowrate as the coolant momentum would resist the crossflow. However, the crossflow intensity increases from Case C00 to Case C26. This is due to the coolant layer along the combustor wall that is present only in Case C26. Based on the velocity contours (fig. 6.3.5), the coolant velocity is slightly lower than the mainstream velocity in the region of high clockwise vorticity (0.4<Y/P<0.5) for Case C26. This means that, compared to Case C00, which has no coolant layer, the horseshoe vortex formed at the leading edge will be stronger for Case C26 (lower velocity near endwall leads to a larger difference between stagnation pressures near the endwall and in the mainstream which, in turn, generates a stronger horseshoe vortex). This means that the passage vortex will also be stronger for Case C26, which is observed in the contour plots by way of stronger clockwise vorticity for this case compared to Case C00. Biesinger and Gregory Smith (1993) [27] also observed that coolant flows with too little momentum can generate stronger secondary flows compared to situations that had no coolant injection.

Streamlines for Cases C00 and C26 have concave curvature when observed from the pressure surface. Case C35 shows such curvature, but the concavity is decreased. Although, like Case C26, Case C35 also has a thick coolant layer along the endwall. The higher momentum of coolant in this case is more successful in resisting migration of the horseshoe vortex from pressure surface to the suction surface. Therefore, endwall crossflow intensity of Case C35 is lower than that of Case C26. Cases C26 and C35 show a region with the strongest negative vorticity near the endwall. These regions are the core of the passage vortex. However, the
locations of these cores are different for the two cases (near $Y/P_i = 0.45$ for Case C26 and near $Y/P_i = 0.3$ for Case C35), with the lower coolant flowrate cases showing the passage vortex to have migrated farther away from the pressure surface. This is, again, the result of higher coolant momentum of Case C35 which resists the cross-pitch pressure gradient and forces the passage vortex core to stay closer to the endwall-pressure-surface corner. Based on the contours and streamline traces, Case C35 does not show strong secondary flow structures. Therefore, it appears to be in the middle of a transition from the conventional passage-vortex system to the impingement-vortex system.

The secondary flows show a different flow structure for Case C46 compared to Cases C00 through C35. The negative vorticity regions in the earlier cases are replaced by mildly positive vorticity (counterclockwise) flow. This happens because the coolant momentum is high enough to have a higher stagnation pressure than the mainstream flow at the leading edge. This creates a counterclockwise flow by which the impingement vortex begins to form. This plane does not show a clear vortex for any coolant flowrates. However, the curvature of the streamlines in Case C46 is convex when looked at from the pressure surface, which indicates a direction of rotation similar to that of the impingement vortex. Case C56 shows even higher positive vorticity and more convex streamlines for Plane 3. The endwall crossflow has ceased to exist in both Cases C46 and C56 due to the proximity of the impingement vortex to the endwall. Similarly, no horseshoe vortex was observed near the endwall-pressure-surface corner. All these observations suggest that Cases C46 and C56 have transitioned to the impingement-vortex secondary flow system.

**Plane 4 (X/Cax = 0.612)**

Plane 4 absolute velocity contours are shown in fig. 6.3.9 while the vorticity plots with secondary flow vectors and streamline traces are presented in fig. 6.3.10 through 6.3.12. Like Plane 3, cases with higher coolant flowrates show higher velocities near the endwall. However, as the flow accelerates from Plane 3 to Plane 4 due the area contraction caused by the vane geometry, the velocities at all locations are increased. Hence, even the lower coolant flowrate cases show less
difference between the velocities near the endwall and in the mainstream at any pitch location, compared to Plane 3 velocity contours. Due to the axial location of this plane (X/Cax = 0.612), there is a strong pitchwise (Y) variation in the velocity with the highest velocity being near the suction surface. The velocity magnitudes are in line with values found in previous studies performed in the same wind tunnel facility (Alqefl et al. (2021) [37-39] and Nawathe et al. (2021) [40,41]).

On Plane 3, Case C00 shows lower vorticity magnitudes along the endwall compared to Case C26. On this plane, Case C00 shows higher vorticity magnitudes. This apparent disparity is an effect of contouring of the endwall. The curvature of the endwall at Plane 3 (X/Cax = 0.289) was highly convex, which led to flow acceleration near the endwall. This acceleration helped in thinning the boundary layer and resulted in a reduction of the secondary flows in the vicinity of the endwall. Unlike at Plane 3, the endwall has becomes concave near Plane 4. This concavity decelerates the near-endwall flow and enhances secondary flows. Therefore, for Case C00, the increase in intensity of secondary flows is expected. For Case C26, this enhancing of secondary flows is partially resisted by the coolant momentum, which was absent in Case C00 and led to a lower clockwise vorticity.
near the endwall. Case C35 provides even more resistance, and the negative vorticity region is forced to remain in the endwall-pressure-surface corner.

Case C26 shows the effects of the passage vortex near $Y/\Pi = 0.7$. Both the vorticity magnitudes and secondary flow streamlines show a strong clockwise curvature to the flow, although a clear center of the passage vortex is not observed. Case C00 does not show a coherent passage vortex structure. But this case has even more intense clockwise vorticity near the endwall. Therefore, it is possible that the strong endwall crossflow dominates this region and the passage vortex effects cannot be observed in isolation. Case C35 shows strong clockwise vorticity near $Y/\Pi = 0.62$, but it is highly concentrated in the endwall-pressure-surface corner. This can be an effect of weakening of the passage vortex and endwall crossflow features due to coolant momentum. As this case is a transitional case.

Figure 6.3.10 Plane 4 vorticity contours with secondary flow vectors and streamlines: Cases C00 and C26
between passage-vortex system and impingement-vortex system, weakening of secondary flow structures is not surprising.

While the pressure leg of the horseshoe vortex evolves into the passage vortex, the suction leg mostly stays along the suction surface for the whole passage. The suction leg has an opposite (counterclockwise) sense of rotation compared to the passage vortex. For Case C00, a small but intense positive vorticity region near Y/\(\Pi\) = 0.9 confirms the presence of the suction leg of the horseshoe vortex. This vortex competes with the endwall crossflow and results in a region of low net vorticity in the spanwise (Z) direction away from the suction leg vortex. Above Y/\(\Pi\) = 0.95, the influence of the suction leg vortex decreases and a clockwise vorticity due to endwall crossflow is seen again. As discussed before, Case C26 generates a stronger horseshoe vortex at the leading edge due to a thicker boundary layer by coolant injection. This means that the suction leg of this vortex will be stronger.

Figure 6.3.11 Plane 4 vorticity contours with secondary flow vectors and streamlines: Cases C35 and C46
than the suction leg vortex observed for Case C00. This is confirmed in the vorticity plots for Case C26. The positive vorticity region is larger, and the vortex is strong enough to nearly completely block the endwall crossflow from reaching the suction surface, unlike in Case C00. For Case C35, Plane 3 results show that the coolant momentum resists formation of a horseshoe vortex. Hence, it is not surprising that, on Plane 4, the effects of the suction leg of the horseshoe vortex are very weak for this case.

The positive vorticity region seen along the endwall in Case C46 is due to the tendency to form the impingement vortex (as explained in the Plane 3 discussion). The streamlines for Case C46 show counterclockwise curvature of the flow near

---

Figure 6.3.12 Plane 4 vorticity contours with secondary flow vectors and streamlines: Cases C56
the endwall. This is similar to that seen in previous impingement vortex studies (Alqefl et al. (2021) [37-39] and Nawathe et al. (2021) [40,41]), albeit a clear center of the impingement vortex does not appear for Case C46. Case C56, on the other hand, shows a proper impingement vortex with its center at $Y/\Pi = 0.62$. Results from the previous chapter (section 5.3) indicate that the impingement vortex moves more away from the pressure-surface-endwall corner with increasing coolant flowrate. When it is sufficiently far away, the crossflow along the endwall reappears. This phenomenon is hinted in the Case C46 contour near the pressure-surface-endwall corner while Case C56 shows significant negative vorticity in this region, indicating the presence of endwall crossflow.

For Cases C46 and C56, weak clockwise vorticity is seen near the endwall-suction-surface corner, which is different than in Cases C26 and C35. This happens because the secondary flow impinges upon the endwall near $Y/\Pi = 0.85$ (see streamline traces for these cases in fig. 6.3.11 and 6.3.12). After impingement, some flow travels toward pressure surface to become a part of the positive vorticity region seen near $Y/\Pi = 0.72$. The remaining flow moves toward the endwall-suction-surface corner, resulting in the negative vorticity region. The thermal concentration measurements taken for Case C56 in the previous chapter (fig. 5.3.12) also confirm that coolant travels toward the endwall-suction-surface corner and then travels away from the endwall along the suction surface, confirming this hypothesis of coolant impingement on the endwall.

**Plane 5 (X/Cax = 1.02, Passage Exit Plane)**

Plane 5 vorticity contours with streamline traces and secondary flow vectors are shown in fig. 6.3.13 through 6.3.15. Cases C00 and C26 show strong endwall crossflow throughout the pitch (Y). Like Plane 4, the flow near the endwall continues to experience deceleration due to the concave endwall contour. Although Case C26 had coolant injection, most of its momentum is lost while traveling through the passage. Therefore, the endwall crossflow is not being resisted and, hence, is most intense on this plane. The higher coolant flowrate cases (C35 through C56) also suffer through a loss of coolant momentum, although the loss is less substantial than in Case C26. Therefore, the endwall
crossflow for these cases is restricted to very close to the endwall, closer than in Cases C00 and C26. Cases C35 and C46 show progressively reducing endwall crossflow compared to C26, but Case C56 shows an increase. In the Plane 4 discussion, it was explained that the impingement vortex moves away from the endwall.
endwall-pressure-surface corner allowing the mainstream flow to become susceptible to endwall crossflow, which exists outside the region of influence of the impingement vortex. Based on these observations, the endwall crossflow has the least influence on the passage exit flowfield when the secondary flow system is in
the middle of transition (Cases C35 and C46). Also, Case C46 shows a positive vorticity region below $Y/\Pi = 1.4$ near the endwall with an increased positive vorticity around $Y/\Pi = 1.15$. This is the core of the impingement vortex, which was
seen on Plane 4 as well. This observation is important because it confirms that Case C46 does not revert to the passage vortex system at any point in the passage. In general, the Plane 5 contours have shown that eliminating endwall crossflow completely at the exit plane is a difficult task irrespective of the secondary flow system that exists in the passage. Case C56 shows a more intense positive vorticity in the same region as that of Case C46, further confirming that the impingement vortex strength increases with coolant flowrate.

Figure 6.3.16 shows the absolute velocity contours for all Plane 5 cases. According to the vorticity contours, the endwall crossflow moves the coolant toward suction surface along the endwall and then along the suction surface away from the endwall. This leads to a drop in velocity near the endwall-suction-surface corner due to mixing caused by endwall crossflow. Based on steady-state mass conservation, the drop in velocity near the suction surface increases the velocity away from the suction surface. Therefore, elevated values of velocities are observed below $Y/P_i = 1.7$. This redistribution of velocity is most visible for Case C00, as the streamlines for this case show the most turning. With increasing coolant flowrate, turning of the streamlines near the suction surface reduces and, hence, flow mixing becomes less intense. This leads to a reduced pitchwise-redistribution of velocity magnitudes for higher coolant flowrate cases. At the same time, increasing coolant flowrate leads to an increase in the total flowrate. Hence, Cases C46 and C56 show overall higher velocities than cases with lower coolant flowrates.
Cases C00 and C26 have exhibited the tendencies of a passage-vortex secondary flow system for the whole passage. The coolant momentum helped in reducing the intensities of some secondary flow features, but these features were never eliminated. Transition from the passage-vortex system to the impingement-vortex system was initiated for Cases C35 and C46. These cases have exhibited features from both secondary flow systems with Case C46 having a nearly complete transition to the impingement-vortex system. Case C56, on the other
hand, displays a proper impingement-vortex system with no passage vortex and a minimal presence of endwall crossflow.

6.4. Conclusions

This study was conducted to document changes in the secondary flow physics due to changes in the flowrate of combustor coolant. A close-coupled combustor-turbine interface geometry was used in this study. As this interface does not feature special geometric features, the results of this study are expected to be applicable for a variety of gas turbine engines. Five cases were considered for this study: four having different combustor coolant flowrates and one without any coolant injection. Five-hole probe measurements were taken at three axial locations to adequately characterize the vane passage secondary flowfield.

The low flowrate cases (C00 and C26) displayed the passage vortex and endwall crossflow to be the most dominant feature in the passage. Cases C35 and C46 proved to be the demarcation flowrates for a transition from the passage vortex secondary flow system to the impingement vortex secondary flow system. The high coolant flowrate case (C56) showed that transition to the impingement vortex system was complete. Overall, the vorticity contours and secondary flow streamlines have shown that the coolant flowrate plays a major role in defining the secondary flow physics.

The injection of coolant provided a resistance to the secondary flow in most cases. Although Cases C35 and C46 did not completely remove the passage vortex system features, they do reduce migration of the passage vortex (or its traces) toward the suction surface and decrease the intensity of endwall crossflow. Decreasing the intensity of these effects is important as coolant skewing toward suction surface is known to deprive cooling to the region of endwall that is near the pressure surface. Case C26 proved to be an exception to this observation. The secondary flows observed in this case were stronger than those observed for no coolant injection (Case C00). This case had a thicker endwall boundary layer due to coolant injection, which resulted in a stronger horseshoe vortex at the leading edge. While other cases with coolant injection also had the same phenomenon,
the momentum of coolant in those cases allowed resisting the stronger secondary flows. For Case C26, this was not possible, which resulted in stronger secondary flows (having clockwise vorticity) than all other cases in this study.

The velocity contours showed that upstream of the exit plane, the absolute velocities away from the endwall were only marginally affected by coolant injection. At the exit plane (Plane 5), the location and intensity of the passage vortex and the endwall crossflow led to a redistribution of velocity magnitudes in the pitchwise (Y) direction with lower velocities near the suction surface and higher velocities away from the suction surface. The redistribution reduces with increasing coolant flowrates as higher flowrates generally tend to result in weaker secondary flows.

The moderate coolant flowrates (C35 and C46) showed no clear dominance of either of the secondary flow systems. This led to the lowest overall vorticity in the passage. The endwall crossflow was never completely eliminated but its intensity was the lowest of these cases. These cases also showed an absence of a coherent passage vortex or impingement vortex. These observations showed that the vortices are easier to eliminate than the endwall crossflow by choosing an accurate coolant flowrate. If designers are interested in reducing secondary flows by coolant injection, combustor coolant mass flowrate ratios between 3% and 5% are expected to be most beneficial.
Chapter 7. Decay of Turbulence in a Vane Passage

Section 7.6.1, the first subsection of section 7.6.2, and section 7.6.3 have been previously presented in Nawathe and Simon (2021) [82]. The relevant figures are cited appropriately but the text in these sections will not be cited again.

7.1. Motivation

The combustor of a gas turbine engine generates a high level of turbulence to create highly effective fuel-oxidant mixing. This turbulent gas mixture travels downstream to the turbine first stage vanes. When experiments involving the turbine vanes were conducted in wind tunnels, the effects of highly turbulent flow on vane surface coolant distributions are observed. However, the evolution of turbulence characteristics in the vane passage was rarely studied in such experiments. As the flow through the vane passage experiences a strong acceleration followed by a mild deceleration, it is important to study how this affects the turbulence. Documenting such measurements was one of the two main objectives of the study that will be discussed in this chapter. After the experimental measurements are recorded, it is valuable to compare such results with those given by computation performed to match the test facility. Specifically, evaluating the performance of computational models that use eddy viscosity turbulence closure models is important as these models are the most used models in the gas turbine industry. Hence, the second objective of this study is to evaluate the vane passage flow using two such models and to compare the computed turbulence characteristics with the experimental results to understand whether the computations can predict such flows with reasonable accuracy.

7.2. Theory of Turbulence

All definitions, expressions and theories presented in this section were taken from Pope (2000) [83] or Tennekes and Lumley (1972) [84], unless stated
otherwise. None of them are the author’s original work. Still, this compilation is presented for the purpose of preparing the reader for the discussion of results.

7.2.1. Characterization of Bulk Flow Properties

Turbulence in the bulk flow is typically characterized by two quantities: the root-mean-square (RMS) fluctuations of the velocity and the turbulence intensity. Velocity measurements with high temporal resolution are taken by a hot-wire probe. They are instantaneous velocities \( u(x, y, z, t) \). First, a mean velocity \( \bar{U}(x, y, z) \) is calculated using the instantaneous velocity and the number of measurements \( N \), as follows:

\[
\bar{U}(x, y, z) = \frac{1}{N} \sum_{1}^{N} u(x, y, z, t)
\]  

(7.1)

Then, the RMS velocity fluctuations \( u'(x, y, z) \) are calculated as:

\[
u'(x, y, z) = \sqrt{\frac{1}{N} \sum_{1}^{N} (u(x, y, z, t) - \bar{U}(x, y, z))^2}
\]

(7.2)

The turbulence intensity \( Tu(\%) \) is defined as the ratio of RMS fluctuations to mean velocity at any location, shown as:

\[
Tu(\%) = \frac{u'(x, y, z)}{\bar{U}(x, y, z)} \times 100
\]

(7.3)

The above quantities are being evaluated for velocity in the primary flow direction, \( u \). In general, three values can be calculated, one for each of the three principal directions.

Before moving further, a few definitions related to specific characteristics of turbulence are introduced:

(a) **Isotropy**: If the RMS velocity fluctuations in each of the principal directions are equal, the flow is said to have ‘isotropic turbulence.’ For many experiments, the turbulence is generated in a way that has it isotropic. Flows not satisfying this condition have ‘anisotropic turbulence.’ Also, many of the models used in computing turbulent flows assume that the turbulence in the entire fluid domain is isotropic.
(b) **Homogeneity**: The homogeneity condition is typically applied to a region or to the entirety of the fluid domain. If the RMS velocity fluctuations (in each direction) at any location in a region of interest are equal, the turbulence is said to be homogeneous. Note that the mean velocity and the instantaneous fluctuations at any two points in the region can still be different. The only quantity that must be matched is the root-mean-square of all velocity fluctuations evaluated over a particular period. This condition is usually achieved in experiments as the flow entering the test facility has uniform mean velocity and the same level of velocity fluctuations. Also, if homogeneity is being defined only for a small region of the fluid domain, it is referred to as ‘local homogeneity.’

(c) **Equilibrium**: A flow is said to have ‘equilibrium turbulence’ if the rate of transfer of turbulence kinetic energy from the larger eddies to smaller eddies is uniform.
This is expected within the inertial subrange of the turbulence spectrum (shown in fig. 7.2.1). Physically, equilibrium turbulence displays a distribution of turbulence kinetic energy amongst different-sized eddies that has a universal form (discussed in section 7.2.2). Usually, turbulence generated in wind tunnels is in equilibrium but if the flow were to undergo strong curvature or acceleration, non-equilibrium turbulence may be generated locally. It follows that in non-equilibrium turbulence, some portions of the inertial subrange will have higher fluctuation levels than others.

7.2.2. Spectral Distribution of Turbulence Kinetic Energy

The bulk flow properties provide only a crude idea about the turbulence in a flow. It is known that turbulent flows have eddies of different sizes that decay to smaller scales and ultimately dissipate by conversion of kinetic energy of turbulence into heat (thermal energy). The frequency associated with any eddy is dependent on its size, with larger eddies occurring at lower frequencies. In any turbulent flow, the kinetic energy of turbulence is distributed among these different sized eddies in a specific manner. This distribution of energy, when plotted against the frequency of eddies is known as the ‘power spectral distribution’ or ‘energy density spectrum.’ For a given mean velocity, the frequency of flow fluctuation level and eddy size can be related to one another with the Taylor hypothesis, frequency = velocity/eddy size. As the flow travels downstream, the energy of the largest eddies is transferred to the smaller eddies, and so on, with the smallest eddies dissipating (i.e., losing their identity) due to viscosity. Due to the nature of this energy transfer, the process of turbulence propagation is referred to as the ‘energy cascade.’

The spectra can be plotted in two ways: one-dimensional or three-dimensional. The three-dimensional spectra plot the distribution of total turbulence kinetic energy, representing the three components of velocity, as a single plot. These are usually difficult to analyze. The other way is to plot the turbulence kinetic energy associated with only a particular component of velocity, typically that of the principal direction. These plots look slightly different than the three-dimensional
plots but are more helpful because it is easier to calculate the scales of turbulence (discussed in the following section). A form of a one-dimensional spectrum for two components of velocity is shown in fig. 7.2.1 as taken from Ames (1994) [85]. The x-axis has wavenumber, which can be directly related to eddy passing frequency, f, while the y-axis has turbulence kinetic energy associated with the one component of velocity, normalized by the freestream velocity. There are three main regions in this energy cascade:

(a) **Energy-containing Range**: The flat region of the plot in the low-frequency range is called the energy-containing range. The largest turbulent eddies belong to this range. As the name suggests, they carry a significant percentage of the turbulence kinetic energy. The sizes of eddies in this range are usually of the same order as the length scale used to characterize the flow. Since the geometry of the flow domain plays a role in defining the sizes of these eddies, flows with similar characteristics (such as jets) but different geometries (such as the jet diameter), have different sizes of the energy-containing eddies. It also follows that the energy-containing eddies can have different sizes in different directions of the same flow (such as with a jet) if the flow has anisotropic turbulence.

(b) **Inertial Subrange**: In the ‘inertial subrange,’ the plots drop down almost linearly (in the coordinates of the figure) with a constant slope (fig. 7.2.1). The eddies in this range are smaller than those in the energy-containing range and receive energy transferred by shear and breakup from the eddies in the ‘energy containing range.’ They are not small enough to dissipate energy significantly by action of viscosity.

An interesting fact about the spectra is that for any flow having equilibrium turbulence, the plot in the inertial subrange is always a straight line having a slope of -5/3 in the coordinates of the plot. In equilibrium turbulence, the energy cascade rate from larger scales to smaller scales within the inertial subrange is essentially uniform over the length scale range. Kolmogorov’s first similarity hypothesis states that for a turbulent flow with sufficiently high Reynolds number, the motion of eddies smaller than the energy-containing eddies has a universal form that can be determined by the kinematic viscosity and the rate at which the energy is
transferred from the energy-containing eddies to the inertial subrange. The latter term has been found to be nearly equal to the rate at which turbulence dissipates in the small scales. It is the rate of dissipation the establishes the rate at which energy cascades down the scales in the inertial subrange. Flows with high energy (high Reynolds numbers) have high energy cascade rates and high dissipation rates in the small scales. As this dissipation rate \( \varepsilon \), in \( \text{m}^2\text{s}^{-3} \), describes the energy cascade, it is a commonly used quantity in both experimental and computational turbulence; thus, further discussions are made only in terms of this quantity. This universality means that the smaller eddies can quickly adapt to the changes that may be happening to the energy-containing eddies due to flow geometry. Even if the flow is anisotropic, the motion of the eddies in this range (as well as the dissipation range to be discussed) can be considered isotropic. In experimental turbulence, achieving a -5/3 slope in the spectrum usually serves as a verification of the accuracy of the measurements.

(c) **Dissipation Range**: For smaller eddies than those of the -5/3 slope of the inertial subrange, the slope of the plot becomes steeper. This is the final region of the plot and is called the ‘dissipation range.’ The eddies in this range are sufficiently small and their velocity gradients are sufficiently steep that they dissipate due to viscosity. As expected, the sizes of the eddies depend on the viscosity of the fluid and the dissipation rate. Like the inertial subrange, the geometry of the flow does not influence the eddies in this region and the eddy sizes in all directions are equal, (i.e., their motion is isotropic).

The high-temporal-resolution velocity measurements are collected using hot-wire anemometry and are used to generate the power spectral distribution. The present measurements are collected at a frequency of 2 kHz for 25 seconds. Previous experiments on the same wind tunnel facility have shown that this collection frequency is sufficient to capture the energy-containing range and the inertial subrange, which are known to carry most of the turbulence kinetic energy. Measurements taken at a single location over a time can be used to generate the spectrum. For the spectrum, the instantaneous fluctuations of velocity, compared
to the mean velocity, $\bar{U}$, must be evaluated first. This is done as shown in the
equation below:

$$u'(t) = u(t) - \bar{U}$$  \hspace{1cm} (7.4)

The next step is to convert velocity fluctuations from time-dependent to
frequency-dependent. This is achieved using Fast Fourier Transform (FFT). The
predefined FFT function in MATLAB is used to perform the analysis. The
transformation is:

$$Q_k = \frac{1}{\sqrt{N}} \left( \sum_{n=1}^{N} u'(t)_k \cdot e^{i \left( \frac{2\pi n}{N} \right) k} \right)$$  \hspace{1cm} (7.5)

Where, $Q_k =$ the transformed array from FFT,
$u'(t) =$ velocity fluctuation relative to the mean,
$N =$ the number of samples of $u'$ over the entire processing time
$k =$ index of $Q_k$ array (k = 1, 2, ..., N/2)
$n =$ index of $u'(t)$ (n = 1, 2, 3, ..., N)

Using parameters based in time domain such as $u'$, the energy density
spectrum, $E(f)$, is defined by Hinze as:

$$\int_{0}^{\infty} E(f)df = u'^2$$  \hspace{1cm} (7.6)

The frequency-dependent array, $Q_k$, is a complex number. Its real part
describes the amplitude of the velocity fluctuation at any index, $k$, while the
imaginary part describes the phase difference between the velocity fluctuation at
index, $k$, and the fluctuation at index 1 (i.e., at time, $t = 0$). Using this transformed
array, the energy density spectrum, $E(f)$, can be defined as follows, using
parameters from the frequency domain:

$$E(f) = \frac{(Q_{\text{real}}[f]^2 + Q_{\text{imag}}[f]^2)}{f_s/2}$$  \hspace{1cm} (7.7)

Where ‘$f_s$’ is the sampling frequency and ‘$f$’ is the fluctuation frequency, defined
as:

$$f = \frac{f_k}{N}$$  \hspace{1cm} (7.8)
The spectral energy, \( E(f) \), is then plotted against the eddy frequency, \( f \), to generate the spectral distribution. The plot shows how much turbulence kinetic energy is present in the flow over various frequency bands, \( \Delta f \), on a log-log scale, like fig. 7.2.1. Recall that the Taylor hypothesis can be used to transform from eddy size to frequency, \( f = \frac{U}{\text{(eddy size)}} \).

### 7.2.3. Length Scales of Turbulence

Once the spectrum is plotted, it can be used to calculate important turbulence length scales. This section will explain the physical significance of each of these length scales as well as the procedure to calculate them from the measurements. It should be noted that all the expressions in this section were derived for homogeneous, isotropic turbulence. As the flow under study may not always satisfy these requirements, the degree of applicability of the length scale expressions will also be discussed in this section.

**Integral Length Scale, \( \Lambda \)**

Integral length scale is a characteristic of the largest eddies (eddies of highest velocity fluctuation magnitude) in the flow. As mentioned before, these have the same scale order of magnitude as the flow scale. To understand how to measure

![Diagram](image)

Figure 7.2.2 A hot-wire probe measures velocities at two spatial locations in a turbulent eddy.
the integral length scale, let us start by considering a single energy-containing eddy in a flow. If we consider any two fluid particles that are travelling with this eddy, their velocities are correlated at a given time. At the same time, if we consider two particles, one inside this eddy and other in a different eddy, their velocities are not correlated as they are influenced by different eddies. So, if we can define the correlation function for all possible distances between any two particles in a flow at any given time, a plot of the correlation function with respect to separation distance can be made. When the value of the correlation is above zero, it can be said that the velocities are correlated. When zero, there is no correlation between the velocities of the two particles. This plot can then be used to calculate a representative length scale.

The above method of correlating the velocities of two spatial locations is known as the ‘two-point correlation’ method. In experiments, taking measurements at two closely-spaced locations at the same time is difficult. Hence, another method of correlation, known as the ‘autocorrelation,’ is used. In this method, two velocity measurements taken at a single location but at two different times are considered. The autocorrelation can be related to the two-point correlation only if we assume that in case of the ‘two-point correlation,’ the mean velocity between the two points does not change while taking the two velocity measurements. This is usually valid if the velocity of the bulk flow is much larger than the velocities of the eddies. This is known as Taylor’s ‘frozen turbulence’ hypothesis. It had been used previously to compute frequency data from length scale data.

Figure 7.2.3 Two-point correlation, $f(r)$, of measurements, presented in [83]
The autocorrelation does not depend on origin of time for “steady turbulence” measurements but rather depends upon the time interval ($\tau = t_2 - t_1$) between the first and the second measurements. The mathematical expression for the autocorrelation ($\rho(\tau)$) between velocity fluctuations ($u'$, as defined in equation 7.2) measured at two different times $t_1$ and $t_2$ is given by:

$$\rho(\tau) = \frac{u'(t_1)u'(t_2)}{u'^2}$$

(7.9)

Where $u'^2$ is the square of the RMS velocity fluctuations defined in equation 7.2. Note that this quantity is not defined for a time as it is a stationary variable and therefore remains the same at any time in a stationary turbulent flow.

At $\tau = 0$, the two velocities being correlated are at the same time and hence the value of $\rho$ will be 1. As the value of $\tau$ is increased, the value of the correlation will go down and reach zero for large $\tau$, which will be the point where the two velocities are no longer correlated with each other. To understand why the magnitude of the expression drops with increasing $\tau$, let us consider how a hot-wire probe sees an eddy that is approaching it, as shown in fig. 7.2.2. Consider time $t_1$ to be when the furthermore point in the eddy reaches the probe. Here, the fluctuation (perturbed value) calculated based on a hot-wire measurement will be $u'(t_1)$ if the eddy does not have net rotation. As the eddy moves downstream, the fluid particles inside the eddy will be detected by the hot-wire probe. Ideally, the measurement taken at the center of the eddy (at time $t_2$) will be equal to the mean velocity of the bulk flow ($\bar{U}$) and $u'(t_2)$ will be very close to zero, again if no net rotation. If substituted in equation 7.9, the value of $u'(t_2)$ will practically come to be very close to zero. Hence, the value of $\rho(\tau)$ will also become zero for the time interval of $(t_2 - t_1)$. If the time interval is further increased, the correlation value can go negative as well. This will happen in an ideal eddy but in a real turbulent flow, the negative values have been observed to remain close to zero. Also, if the time interval is too large, the concept of the correlation function, $\rho(\tau)$ does not hold. The above explanation shows that the autocorrelation is indeed a measure of how velocities at two different times are correlated. If the 'frozen turbulence' hypothesis holds, a simple
relation between the time-based autocorrelation function \(\rho(\tau)\) and the spatial two-point autocorrelation function \(\rho(x)\) can be made as follows:

\[
\rho(x) = \rho(t) \ast \bar{U} 
\]  
(7.10)

Figure 7.2.3 shows a representative spatial correlation function plotted against a non-dimensionalized distance (non-dimensionalized by the mesh size of the turbulence generator) between two particles. After that, Taylor defined a length scale that is equal to the area under the curve shown in the figure:

\[
\Lambda = \int_0^\infty \rho(x) \, dx 
\]  
(7.11)

This quantity is termed to be the ‘integral length scale’ by Taylor as it is integrated for all possible distances between any two particles in the flow. He said that the value of this quantity can be regarded as the ‘average size of the eddies.’ On the other hand, Pope [83] calls it ‘a characteristic of larger eddies’, although he does not provide an explicit reason for this statement. As Pope’s definition is commonly used in recent literature, it is assumed henceforth that the integral scale is representative of the larger turbulent eddies. The integral length scale can be calculated for each of the three principal directions by defining the correlation functions for the respective components of the velocities. Hinze (1975) [76] used an alternative approach for describing the integral length scale from a curve-fit expression for the power spectral distribution of a typical homogenous, isotropic turbulence (like the one plotted in fig. 7.2.1). This requires a considerable amount of data for characterizing the large-scale part of the spectral turbulence kinetic energy curve. This alternate form of the integral length scale calculation, given by Hinze, is shown below:

\[
\Lambda = \frac{\bar{U}}{4} \ast \lim_{f \to 0} \frac{E(f)}{u'^2} 
\]  
(7.12)

While both equations 7.11 and 7.12 give the same result, the latter equation is much easier to compute for experimental studies once the Fourier transform is computed. Hence, this equation is used to calculate the integral length scales for the current study. Note that the spectral energy term in equation 7.12 corresponds to only very low frequencies (the limit as \('f'\) tends to zero). This means that only
the energy-containing range of the spectrum should be used to evaluate \( E(f) \).

Based on the spectra generated for this study (to be discussed in section 7.5.2), the frequencies below 1 Hz are considered while evaluating the integral length scale. However, the number of data points for such small frequencies for this study is low. Additionally, it is possible that the fluctuations measured by the probe for these frequencies do not relate to turbulence, precisely, but also could relate to unsteadiness present in the wind tunnel. Therefore, the integral length scale calculated from these data involve significant uncertainty. As discussed before, the integral length scale depends on the flow geometry. So, its value will change based on where the measurement is taken. Therefore, a thorough documentation of the spatial variation in this length scale is provided in the results section. Finally, equation 7.12 was given for homogeneous, isotropic turbulence and therefore may not apply well to flow in the latter half of the vane passage that has experienced strong acceleration, some deceleration and strong curvature. Nonetheless, the change in the values of the integral length scales through the vane passage may still give an idea about the variations and evolution of the largest eddies in the flow.

**Taylor Microscale, \( \lambda \)**

Taylor was able to prove that the two-point correlation function, \( \rho(x) \) can be related with the rate of decrease of the velocity in the direction under consideration [86]:

\[
\left( \frac{\partial u}{\partial x} \right)^2 = 2u'^2 \lim_{x \to 0} \left( \frac{1 - \rho(x)}{x^2} \right)
\]  

(7.13)

He then defined a length scale, \( \lambda \), that is related to the rate of decrease of the velocity and represents the 'diameter of the smallest eddies, which are responsible for dissipation of energy.' This length scale, which is now known as the 'Taylor microscale,' is calculated using the following expression:

\[
\frac{1}{\lambda^2} = \lim_{x \to 0} \left( \frac{1 - \rho(x)}{x^2} \right)
\]  

(7.14)

In the same paper, he also showed that turbulence dissipation, \( \varepsilon \), is dependent only on the kinematic viscosity \((\nu)\) and the instantaneous velocity distribution. For isotropic turbulence, he developed the following expression for the dissipation:
\[ \varepsilon = \frac{15\nu}{2} \left( \frac{\partial u}{\partial x} \right)^2 \]  

(7.15)

Combining equations 7.13 through 7.15 leads to an expression for the Taylor microscale in terms of the turbulence dissipation. This expression, like the integral length scale expression in equation 7.12, does not require calculation of the correlation function and, thus, is easier to apply. This expression is:

\[ \lambda = \frac{15 \nu * u'^2}{\varepsilon} \]  

(7.16)

Taylor's assumption that the above microscale represents the smallest eddy size in the flow was later proven to be incorrect by Kolmogorov. Kolmogorov attributed the error to the fact that Taylor considered \( u' \) to be the characteristic velocity of the dissipative eddies. In fact, the Taylor microscale represents the eddies that belong to higher frequency end of the inertial subrange of the turbulence spectrum where eddies are on the verge of dissipating. The Taylor microscale can be calculated with relative ease and certainty; hence, is still used by researchers to characterize turbulence.

**Kolmogorov Scale, \( \eta \)**

Kolmogorov hypothesized that the smallest scales of turbulence are universal and are dependent only on kinematic viscosity, \( \nu \), and turbulence dissipation, \( \varepsilon \). If these quantities are related in a way to derive a dimensionless constant from them, that relation will be universal. But, these two quantities, by themselves cannot form a dimensionless quantity. Hence, Kolmogorov defined a length scale that was equal to a relationship among the above two quantities. This length scale is now known as the 'Kolmogorov scale,' \( \eta \), and is expressed as below:

\[ \eta = \left( \frac{\nu^3}{\varepsilon} \right)^{1/4} \]  

(7.17)

Calculation of the turbulence dissipation, \( \varepsilon \), in the dissipation range involves high uncertainty owing to the high-frequency measurements which, if the sampling is not sufficiently fast, tend to suffer aliasing error. However, the dissipation rate calculated using the inertial subrange part of the spectrum is equal to the
dissipation calculated in the dissipation range. With that, the uncertainty in magnitude of this length scale need not be very high despite not having good resolution of high-frequency measurements.

The turbulence length scales that have been discussed up to this point belong to specific regions on the power spectral distribution; in that, they represented ranges of eddy sizes. Roughly speaking, the integral length scale corresponds to (1) the eddies in the high-amplitude portion of the spectrum, referred to here as the energy-containing range and (2) the large-eddy portion of the inertial subrange. The Taylor microscale corresponds to the eddies that are near the small-eddy portion of the inertial subrange that are near the dissipation range while the Kolmogorov scale represents the smallest eddies that are dissipating due to viscosity and high shear rate.

Other Scales of Turbulence

In experimental turbulence, in addition to the length scales discussed above, one more length scale is used but its definition and physical representation is not consistent among different sources. This is referred to as the ‘energy length scale’ by the present author. Each of the definitions of this scale are given below:

(a) Tennekes and Lumley (1972) [84]: Using the description of turbulence dissipation originally given in Taylor (1935) [51], the expression for energy length scale is given as:

\[ L = \frac{u'^3}{\varepsilon} \]  \hspace{1cm} (7.18)

No physical explanation of this quantity is given in Tennekes and Lumley.

(b) Pope (2000) [83]: Pope gives the energy length scale to establish a relationship between the Taylor scale and Kolmogorov scale. He states that this length scale is a characteristic of the larger eddies but does not state why it is different than the integral length scale. The expression for this scale is:

\[ L = \frac{k^{3/2}}{\varepsilon} = \left(1.5^{1/2}\right) \left(\frac{3}{2} \times \frac{u'^3}{\varepsilon}\right) \]  \hspace{1cm} (7.19)

Where \( k \) is the turbulence kinetic energy, which is equal to \( 1.5 u'^2 \) for isotropic turbulence.
(c) Hancock and Bradshaw (1989) [87]: They defined the energy length scale to compare turbulence generated in different experiments. The expression for the length scale is as follows:

$$L = \left( \frac{3}{2} \frac{u'^3}{\varepsilon} \right)$$

(7.20)

(d) Ames and Moffat (1990) [77]: Their expression of the energy length scale is the same as that of Hancock and Bradshaw. Furthermore, they give an expression (discussed later) for turbulence dissipation based on the turbulence energy in the inertial subrange. They claim that this definition of dissipation makes the energy length scale have a clear relationship with the power spectral distribution. As the turbulence generator used in the current facility is built based on the recommendations of the first author of this reference, Ames, the definition and expression of the energy length scale given in this reference are used in the present study. In this way, a comparison between the turbulence generated in the current facility and in Ames’ facility can be made.

To calculate the energy length scale using the definition of Ames and Moffat, the turbulence dissipation \( \varepsilon \), must be defined. Their paper states that this value can be calculated using the inertial subrange of the power spectral distribution. In this region, it is known that the distribution has a constant -5/3 slope in the coordinates of the plot (Figure 7.2.1). They gave the following relation between the turbulence kinetic energy \( E(f) \) and frequency \( f \):

$$E(f) = \frac{18}{55} A \varepsilon^{2/3} f^{-5/3} \left( \frac{2\pi}{U} \right)^{-2/3}$$

(7.21)

As the power spectral distribution is plotted on a log-log scale, let us take the log on both sides of this equation:

$$\log(E(f)) = -\frac{5}{3}\log(f) + \log\left( \frac{18}{55} A \varepsilon^{2/3} \left( \frac{2\pi}{U} \right)^{-2/3} \right)$$

(7.22)

Where ‘\( A \)’ is a constant and equal to 1.62, according to Ames and Moffat [77].

As the above equation is an equation of a line in the form ‘\( y = m x + c \)’, any one combination of \( E(f) \) and \( f \) can be used to evaluate the dissipation, \( \varepsilon \). After
finding the dissipation, equation 7.20 can be used to calculate the energy length scale.

This section explained the experimental part of the theory of turbulence. The field of turbulence modeling is not discussed in detail as it is vast. The most pertinent details of the fundamentals of turbulence modeling are briefly touched upon in section 7.5.1, where the turbulence models used in this study are discussed.

7.3. Experimental Procedure

The first part of this study involved taking velocity measurements at different locations throughout the vane passage. These measurements help in explaining how various turbulence features change in the passage. Five axial (X) locations (called planes) were chosen inside the passage. These locations correspond to locations of the planes discussed in the earlier studies. The same locations were chosen as they seem to provide good characterization of the passage flow, and the data acquisition system was already set up to take measurements at these locations. Figure 7.3.1 shows the locations of the planes. Additionally, an axial location farther upstream (X/Cax = -0.81) was added for two purposes. The first is that it verifies that the turbulence features of the flow entering the test section are representative of a combustor exit flow. The second is that the measurements taken at this location serve as the inlet boundary conditions for the numerical simulations.
On each of the planes, suitable locations are needed for measurements. As the goal of the experiment is to achieve results that can apply to a general vane passage flow, the measurements are taken near the center of each plane. Based on the aerodynamic measurements presented in Chapter 5 and 6, there are no major secondary flows on the selected planes at those locations. Hence, velocity and turbulence measurements recorded at these locations will not be influenced by endwall shape, coolant flows, or vane shapes, other than their effects on the core acceleration. Figure 7.3.2 shows the measurement locations for each of the planes. Note that the axes use local pitch (P_{local}) and span (S_{local}) values to nondimensionalize pitch (Y) and span (Z) locations. Twenty-five measurement points per plane were recorded in the passage, except for plane 4 where fifteen measurement points were recorded. For the approach plane (X/Cax = -0.81), the measurements for the entire pitch and span were taken as part of an earlier study and are presented in section 4.3.

Figure 7.3.1 Axial locations for measurements inside the vane passage [82]
Single-wire hot-wire anemometry was used to measure velocities sampling at 2000 Hz for 25 seconds. Details of the workings of a hot-wire probe are given in section 3.6. while the uncertainty calculations are presented in Appendix A. The hot-wire probe is expected to measure the streamwise component of velocity for each location. The five-hole probe measurements taken in an earlier study provide the angle of the streamline at the center of each plane. This angle is used to correctly orient the measurement plane of the hot-wire probe. The uncertainty in this angle (based upon the uncertainty of the five-hole probe) is expected to be
lower than 3° and is sufficiently low for accurate characterization of streamwise
turbulence.

7.4. Numerical Simulations Procedure

In the gas turbine industry, RANS models are most widely used. These models
do not require large computation time (as compared with Large Eddy Simulations)
and hence, are useful for solving complex fluid mechanics problems. However, as
the turbulence features are modeled instead of being calculated from basic
principles, these models tend to have some inaccuracy associated with them.
Based on the work of El-Gabry et al. [56], the k-ω models are expected to give the
best results for RANS modeling of gas turbine flows. Hence, two types of the k-ω
models, the Standard and the Shear Stress Transport (SST) models, were used
to perform simulations. A discussion of these models is first given, followed by an
explanation of the numerical setup of the study.

7.4.1. Turbulence Closure Models

As the Navier-Stokes equations are difficult to solve in their original form, a
technique known as ‘Reynolds Averaging’ is used to decompose quantities
(velocities and pressure terms in case of the Navier-Stokes equations) into
average components and fluctuating components. Performing this decomposition
on the Navier-Stokes equations results in the Reynolds-Averaged Navier-Stokes
(RANS) equations. The RANS equations are:

\[
\frac{\partial (\rho \bar{u}_i)}{\partial t} + \frac{\partial (\rho \bar{u}_i \bar{u}_j)}{\partial x} = - \frac{\partial P}{\partial x_i} + \frac{\partial (\tau_{ij} - \rho \bar{u}_i \bar{u}_j)}{\partial x_j} + \rho g_i
\]  

(7.23)

Where \( \rho \) is the density, \( u_i \) is the velocity in the ‘i’ direction, \( P \) is the pressure and
\( \tau_{ij} \) is the stress tensor. The overbar (e.g., \( \bar{u}_i \)) represents a time-average term while
a prime (e.g., \( \bar{u}_i \bar{u}_j \)) represents a fluctuating term. The equation shows that the
fluctuating terms are present only in the term \( \rho \bar{u}_i \bar{u}_j \), which is known as the
‘Reynolds Stress’ term. While most of the average components can be evaluated,
the same is not possible for the fluctuating components. Hence, the Reynolds
stress term must be modeled to solve the RANS equations. As a way of simplifying,
the Reynolds stress term (as given by Boussinesq in 1877) can be described as follows:

$$-\rho u'_i u'_j = \rho E_{ik} \left( \frac{\partial \bar{u}_k}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_k} \right)$$  \hspace{1cm} (7.24)

Where $E_{ik}$ is called the eddy diffusivity. The Reynolds stress (a term containing fluctuating terms) is defined by using the averaged terms on the right-hand side. In this way, the need to solve for the fluctuating terms is eliminated, albeit at the cost of accuracy. Clearly, this is a modeling of the Reynolds stress term and is no better than the eddy diffusivity term, $E_{ik}$. The equation now contains the new unknown, eddy diffusivity. The eddy diffusivity can be modeled in multiple ways, and this gives rise to different types of turbulence closure models that are seen in computational fluid mechanics problems. These are collectively referred to as the 'RANS Turbulence Closure Models.'

The k-$\omega$ model is one such way to define the eddy diffusivity. In this model, the eddy diffusivity is solved by writing two additional transport equations, one each for turbulence kinetic energy, $k$ (m$^2$.s$^{-2}$) and specific dissipation rate, $\omega$ (s$^{-1}$). The specific dissipation rate, $\omega$, is defined as:

$$\omega = \frac{\varepsilon}{C_{\mu}k}$$  \hspace{1cm} (7.25)

Where '\varepsilon' is the rate of dissipation of turbulence (m$^2$.s$^{-3}$) and $C_{\mu}$ is a model constant that typically has a value of 0.09. The most important assumption made in this model is that, although the eddy diffusivity is a tensor, it is assumed to be a scalar. This assumption automatically makes the turbulence of any flow being solved isotropic. This will play a role when the results of this study are discussed.

The k-$\omega$ models give good results in the boundary layer of a flow having an adverse-pressure gradient when compared to the k-$\varepsilon$ models, which give poor results near walls. As mentioned earlier, there are two different subtypes of the k-$\omega$ models. The 'standard' model solves the equations that are presented as the original k-$\omega$ model in Menter (1994) [88]. However, it was found that even minute changes in the mainstream turbulence intensity significantly change the results of a standard k-$\omega$ model. In other words, the solution may not be very stable if
external flows are being evaluated. To counter this, the shear stress transport (SST) k-ω model was introduced by Menter (1994) [88]. The SST model solves the k-ω equations only near the wall and solves the k-ε equations away from the wall, then imposes blending of the solutions of these two equation systems. As the k-ε model is solved in the freestream, the turbulence intensity of the wall-bounded flow does not destabilize the equation. At the same time, as the k-ω model is solved near the wall, relatively accurate predictions of the boundary layer flow are achieved. The k-ε models require very fine meshes near walls for good solutions, which leads to an increase in computational time. Hence, only the standard k-ω and the SST k-ω models were chosen for the present study. As the SST model reverts to the k-ε equation system in the mainstream, the present results applied to the mainstream are expected to be identical to the k-ε model results. A third model, known as the Spalart-Allmaras (SA) model was initially considered. However, it was found that, due to the nature of the model, its results could not readily be compared with the solutions of the other models or with the experimental measurements. Hence, the computational runs of the SA model were not completed, and the results are not documented here.

7.4.2. Simulations Setup

ANSYS Fluent 18.2 is used to perform the simulations. The fluid domain is shown in fig. 7.4.1. The domain inlet is same as the experimental approach plane (X/Cax = -0.81). The domain exit is located at X/Cax = 1.35. The exit was kept sufficiently downstream of the vane passage exit to prevent the computations from generating any artificial backflow to prevent convergence. ANSYS Mesh was used to generate an unstructured mesh for the domain. To test for grid independence, the SST k-ω model case was run with three different mesh sizes. They resulted in 0.6, 1.5 and 2.8 million nodes. As the turbulence features are important for this study, values of the velocity fluctuations calculated by different meshes were chosen to compare the results with various meshes. Figure 7.4.2 shows the line plots of these values. The performances of all three cases are nearly identical. Based on computation time, the mesh with 1.5 million nodes was chosen for
running the remainder of the simulations. Although a fine mesh was used only near some of the walls, the k-ω models do not require such meshing if boundary layer fluid physics is not being studied. As all relevant computational results are in the center part of each plane, the mesh size should not affect the solutions. A steady simulation is run by using the ‘coupled’ solver provided in ANSYS Fluent. The boundary conditions used for simulations are tabulated in table 7.4.1. The inlet plane values are measured in the test facility while the exit plane values are expected to be equal to atmospheric pressure, as the plane is located far downstream of the vane passage. As the flow simulations are complex, convergence was assumed when the residuals for velocities and turbulence parameters were below $10^{-4}$ and the residuals for continuity were below $10^{-3}$. The residual values of all parameters were found to be plateauing when plotted on a log-log scale.

Figure 7.4.1 Computational fluid domain [82]
As RANS models do not calculate fluctuating components of velocity, an equivalent fluctuation term must be derived from the solutions of the simulations. The simulations provide the values of turbulence kinetic energy, $k$, at all locations. This energy can be related with velocity fluctuations using the following relation:

$$L'_i = \frac{2}{3}k$$ (7.26)

As RANS models assume that the turbulence kinetic energy is equal in all directions (isotropic turbulence assumption), the value of $u'$ is considered to represent velocity fluctuations in any direction. As the average velocity is calculated in the simulations, an equivalent turbulence intensity can also be calculated using equation 7.3. The experimental integral length scales are calculated using the power spectral distribution. As the computations do not give such distribution, equivalent integral scales cannot be calculated. On the other hand, equivalent Taylor microscales, Kolmogorov scales, and energy scales can
be calculated by using the expressions discussed earlier (equations 7.16, 7.17, and 7.20).

Table 7.4.1 Boundary conditions for numerical simulations

<table>
<thead>
<tr>
<th>Location</th>
<th>Conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet Plane</td>
<td>Uniform Axial Velocity, $u = 12.4$ m/s</td>
</tr>
<tr>
<td></td>
<td>Turbulence Kinetic Energy, $k = 2.05$ m$^2$.s$^{-2}$</td>
</tr>
<tr>
<td></td>
<td>Specific Dissipation Rate, $\omega = 224.13$ s$^{-1}$</td>
</tr>
<tr>
<td>Exit Plane</td>
<td>Uniform Pressure, $P = 0$ Pa (gauge)</td>
</tr>
<tr>
<td>Top and Bottom Walls</td>
<td>Periodic</td>
</tr>
<tr>
<td>All Other Walls</td>
<td>No-slip</td>
</tr>
</tbody>
</table>

7.5. Results and Discussion

As mentioned earlier, measurements were taken at multiple locations on each plane, in the central part of that plane. The reason for taking multiple locations was to validate that the bulk flow properties of turbulence do not change from point to point. Once that was established for most of the line plots presented later, only one of locations on each plane was chosen to represent that plane. As mentioned before, fig. 7.3.2 shows where the measurements were taken for planes 1 through 5. The solid black lines in each plot indicate the boundaries of the planes and the measurement locations are indicated by black square points. Note that the axes are normalized by the local pitch ($Y$) and span ($Z$) values to ensure that the aspect ratio of the plane remains the same and that the measurements are taken in the same general area on each of the planes. Measurements were taken only for $Z/S_{\text{local}} > 0.5$ to ensure that no contoured endwall and film cooling effects are present. Similarly, a minimum clearance of $Y/P_{\text{local}} = 0.2$ was kept from both the suction surface and pressure surface to ensure that wall effects do not significantly affect the measurements. Measurements for the approach plane were taken for the whole plane and were presented in section 4.3.
The mean velocity contours for all measured planes are shown in fig. 7.5.1. For planes 1 through 4, the mean velocity distributions show little variation in the spanwise (Z) direction, but a larger velocity is observed nearer the suction surface (Y/P_{local} = 1). This is expected as the flow accelerates near the suction surface owing to its convex shape. For plane 5 (passage exit), the velocity contours show uniform velocity everywhere. This is correct as the flow exiting the vanes is to have a flat velocity profile. Therefore, a preliminary analysis of these contours shows that the velocities in the passage are as they should be.

The RMS velocity fluctuations for all planes are shown in fig. 7.5.2. For planes 1 through 4, the differences in fluctuation levels along any plane are minimal. As the fluctuations are one of the key characteristics when turbulence is compared at
a bulk scale, matching these values across a plane implies that the streamwise turbulence over the central part of any axial plane in the passage is uniform. For plane 5, measurements in the top-right corner show a rise in velocity fluctuations while the rest of the plane shows uniform values. These unexpected fluctuations are believed to be a result of interference with the flow in the adjacent passage. Hence, these measurements are ignored and the measurements of the remainder of the plane are held up for discussion. The turbulence intensity for all planes is plotted in fig. 7.5.3. Like the velocity fluctuations, the contours on each plane show only minor changes in their values throughout the measured zones (except for the corner of plane 5 discussed above). This shows that any one of the measurements for each plane can be chosen for the discussion of bulk properties. For each plane,
the location with $Y/P_{\text{local}} = 0.5$ and $Z/S_{\text{local}} = 0.65$ was chosen as representative of that plane. The reasons for choosing this location are given below:

(i) The location is one of the locations that is farthest from any of the walls.

(ii) The power spectral distribution for all planes at this location show the least number of fluctuations after filtering the data.

![Figure 7.5.3 Turbulence intensity contours](image-url)
Many of the plots in the following sections feature both computational and experimental results. The experimental results were taken at six axial locations. The computational results are also plotted for the same six locations. Additionally, three more locations between the approach plane (X/Cax = -0.81) and plane 1 (X/Cax = -0.02) were chosen for the computational results to get better trends for the flow approaching the vane passage.

7.5.1. Bulk Flow Properties

Figure 7.5.4 shows a line plot of mean velocities through the vane passage. The experimental measurements at the approach plane show a velocity of 12.47 m.s\(^{-1}\), which was used as one of the boundary conditions at the inlet of the computational domain. As the flow enters the passage at plane 1 (X/Cax = -0.02), the velocity has increased to about 20 m.s\(^{-1}\). Downstream, the flow accelerates to 23.4 m.s\(^{-1}\) at plane 2 (X/Cax = 0.289). Up to this location, the computational results agree well with the experiments. The flow then experiences strong acceleration due to the decrease in the flow cross-sectional area. At plane 3 (X/Cax = 0.612), the experiments show a velocity of about 46 m.s\(^{-1}\). The computations slightly overpredict this velocity to give a value of 49 m.s\(^{-1}\). When the flow has reached the throat of the passage at X/Cax = 0.775 (plane 4), the highest velocity in the passage is reached. Both the experiments and computations show this value to be
around 59 m.s\(^{-1}\). Downstream of the passage, the flow starts to decelerate due to increasing cross-section area and exits the passage at 58 m.s\(^{-1}\) for the experiments and 56 m.s\(^{-1}\) for the computations. Generally, the computations and experiments show similar mean velocities through the passage and the trend observed in the plots is typical for a high-pressure turbine vane. Hence, this indicates that both the experiments and simulations were run correctly.

The RMS velocity fluctuations in the vane passage are shown in fig. 7.5.5. It is worth pointing out again that the hot-wire measures only the streamwise component of the velocity and, hence, can be used to calculate only the streamwise velocity fluctuations. For computations, the RANS models assume isotropy and, hence, will effectively have the same level of velocity fluctuations in any direction, including the streamwise direction. Therefore, the plots in the figure correspond to streamwise velocity fluctuations. At the approach plane, the fluctuation level of 1.2 m.s\(^{-1}\) was recorded. In the experiments, as the flow accelerates downstream, the velocity fluctuations decrease to 0.9 m.s\(^{-1}\) at the passage inlet (plane 1). The fluctuations decrease further at a faster rate as the acceleration increases inside the passage to 0.83 m.s\(^{-1}\) at \(X/C_{ax} = 0.289\) (plane 2) and to 0.65 at \(X/C_{ax} = 0.612\) (plane 3). They reach a minimum of 0.62 m.s\(^{-1}\) at the throat plane \((X/C_{ax} =0.775)\). As the velocity fluctuations are directly proportional to the level of turbulence in a flow, a decline in their values clearly shows that the streamwise turbulence is decaying in the vane passage. The slope of the curve also gives an idea about the rate of this decay. This decrease in fluctuations is caused by stretching of the turbulent eddies in the streamwise direction due to the strong acceleration in that direction. The acceleration continues to the throat of the passage and is followed by a mild deceleration as the passage exit is approached. This deceleration leads to the rise of velocity fluctuations from 0.62 m.s\(^{-1}\) at the throat of the passage to 0.97 m.s\(^{-1}\) at the exit. Although the deceleration is mild and occurs for a shorter distance than the acceleration upstream, the compressive strain destabilizes the flow significantly, leading to a rapid increase in velocity fluctuations.
The computational results of the velocity fluctuations show a very different trend compared to the experimental measurements. For both models, the inlet boundary condition for the velocity fluctuations are equal to the value measured in the experiments (1.2 m.s\(^{-1}\)). However, between the approach plane (X/Cax = -0.81) and the passage inlet plane (X/Cax = -0.02), the velocity fluctuations remained practically unchanged. Inside the vane passage, the velocity fluctuations increase with acceleration of the flow for both models. At the throat plane (X/Cax = 0.775), where the flow is experiencing the highest acceleration and highest velocity, the fluctuations predicted by the k-ω standard model are 2.03 m.s\(^{-1}\) and by the k-ω SST model are 1.40 m.s\(^{-1}\), compared to the value of 0.62 m.s\(^{-1}\) in the experiments. Downstream of the passage throat, where the flow is decelerating, the k-ω SST model shows a decrease in fluctuation levels while the k-ω standard model continues to show an increase in fluctuations. This disparity in trends of the fluctuation levels between the models and experiments is suspected to be due to the assumption of isotropic turbulence made in the RANS models. As shown by Taylor (1935) [51] and Vicharelli and Eaton (2006) [55], in an accelerated flow, the velocity fluctuations in the streamwise direction decrease while those in the cross-stream direction can increase. As the RANS models assume a single value for
eddy viscosity in all directions, it effectively neglects the effect of acceleration in a particular direction on turbulence. In a sense, these models are averaging out the fluctuations in all directions for anisotropic flows and thus show higher values of velocity fluctuations than the experimental results. This increase is incorrect. This shows that computations of the mainstream turbulence levels of a vane passage flow using RANS models will always deviate from the experimental results.

The turbulence intensity line plots are shown in fig. 7.5.6. The measurements recorded the highest turbulence intensity of 9.5% at the approach plane. It reached a minimum to about 1% at the passage throat. At the passage exit, deceleration has caused the turbulence intensity to rise back to 1.5%. The computations also show a similar trend of turbulence intensities when compared to the experiments, albeit with higher magnitudes. The earlier discussions, however, have shown that the computations do not produce results like the measurements. This discrepancy can be explained by looking at the expression of the turbulence intensity (equation 7.3). The velocity fluctuations in the numerator are incorrectly predicted by the computations while the mean velocities in the denominator are generally well predicted by the computations. Because the magnitudes of the mean velocities are much higher than the velocity fluctuations, the values of the computed turbulence intensities are influenced mostly by the mean velocities. Therefore, these values create an illusion that the computations of turbulence intensity are showing a good performance when compared with the experiments. Many studies in the literature assume that matching the experimental turbulence intensities to their modeled results is an indication of a good model. However, the current study shows that to verify a model, the RMS velocity fluctuations (i.e., turbulence level) should be compared with the experimental results. Alternatively, as the velocity fluctuations are calculated using the turbulence kinetic energy, $k$, a comparison of experimental and computational turbulence kinetic energy must be made for model verification.
7.5.2. Power Spectral Distribution

Axial Variation in Spectral Distribution

To study how the turbulence kinetic energy distribution changes in the vane passage, the power spectral distribution was generated for one location per each plane for all the planes. These locations are the same as those used to discuss the bulk flow properties. All spectra are plotted on a log-log scale in fig. 7.5.7. For the approach plane, the spectrum looks like those observed in typical equilibrium turbulent flows. The flow accelerates mildly up to plane 2 (X/Cax = 0.289), and the shape of the distribution is unaffected. The magnitudes of the fluctuation levels contained in the low-frequency region decrease considerably, but this is expected, as the turbulence is decaying. This means that the net content of the turbulence kinetic energy decreases. The flow downstream of plane 2 starts to accelerate considerably and this effect is seen in the spectral distributions, which deviate significantly from its upstream shape. The low-frequency eddies continue to show similar levels of energy content but the eddies at the low-frequency end of the inertial subrange experience significant drops in their fluctuation levels. This happens because these eddies are getting stretched and reduced in size due to acceleration. This stretching also leads these eddies to transfer their energies to smaller eddies at a faster rate than the rates seen upstream of these locations. Hence, the eddies near the high-frequency side of the inertial subrange and in the dissipation range for planes 3 through 5 have similar levels of turbulence fluctuation content as the eddies with the same frequencies in the upstream part of the passage. Interestingly, the -5/3rd slope for the spectrum in the inertial subrange is observed for all planes. However, owing to flow acceleration, the location of the low-frequency portion of the -5/3 range changes through the passage. The reader can refer to fig. 7.5.8 for a closer look at the inertial subrange for all spectra. For all planes, the shapes of the curves for frequencies beyond this range have reduced slopes. This may be an effect of lower fluctuation level and an influence of aliasing error. The sampling frequency for the measurements is 2000 Hz and, hence, by the Nyquist–Shannon sampling theorem, any data for frequencies above 1000 Hz will have aliasing error. Practically, the aliasing can
appear in frequencies as low as 700 Hz in this low-fluctuation-level curve. Hence, only the data for frequencies less than 750 Hz are considered to have a low enough uncertainty for discussion.

Pitchwise Variation in Spectral Distribution

The spectra presented in the earlier subsection were plotted for only one location per plane. The reasons behind the choice of this location have already been discussed. However, the way in which the spectral distribution changes when one moves from the pressure surface to the suction surface is not known. Such variation is discussed in this subsection. Plane 3 is chosen for this discussion as this plane shows the largest change in mean velocity magnitude in the pitchwise (Y) direction and the flow at this location is significantly affected by shapes of both the vane pressure surface and suction surface. Also, two different spanwise locations are chosen for discussion to identify whether there is significant variation of the spectra in the spanwise (Z) direction. These locations are \(Z/S_{\text{local}} = 0.5\) and 0.664 (refer to fig. 7.3.1). The spectra for each spanwise location are shown in fig. 7.5.9 and 7.5.10. Each measurement location is identified based on its pitch and span co-ordinates and is shown alongside the spectra. Based on the RMS velocity fluctuations (fig. 7.5.2), the total turbulence kinetic energy in the flow at any location

![Figure 7.5.7 Power spectral distribution](image1)

![Figure 7.5.8 Power spectral distribution (inertial subrange)](image2)
on plane 3 does not change significantly. But the spectra show that the distribution of energy among the different eddy sizes changes with location. For both spanwise locations, the measurements near the suction surface (locations 5.1 and 5.3) show higher energy content in the low-frequency region (<10 Hz) than shown in the measurements near the pressure surface (locations 1.1 and 1.3). This is possibly because of the change in pressure field due to curvature of the streamlines as directed by suction surface curvature. At plane 3, the wall curvature is beginning to become concave, which is known to lead to a destabilization of the pressure field and generation of unsteadiness. This unsteadiness usually is observed in the low-frequency part of the spectrum. This is observed in the hot-wire probe data as a rise in low-frequency energy content for locations 5.1 and 5.3. On the other hand, energy content in the inertial subrange (100-400 Hz) is higher for locations near the pressure surface (locations 1.1 and 1.3). Such increase is observed when the flow acceleration stretches the largest eddies and energy is transferred from the larger eddies to those in the inertial subrange. This effect is at a faster rate than is observed for equilibrium turbulence. As the flow accelerates near the pressure surface, this may have led to the rise in energy within the inertial subrange for locations 1.1 and 1.3. For eddies at higher frequencies than 400 Hz, the spectra seem to merge, implying that the energy distribution among these eddies is unchanged. This is expected as smaller eddies are known to typically not be affected by changes in the bulk flow properties.

Although the spectra in fig. 7.5.9 and 7.5.10 show the energy distribution (the amplitude of oscillation), it is important to note that both amplitude and frequency describe the energy contained in the eddies and, hence, the spectra are not good measures of the rate of energy convected in the flow at that location. To understand the contribution of each eddy size to total energy in the turbulence, the spectra are modified. Specifically, the Y-axis quantity is changed from spectral energy (or amplitude), $E(f)$, to a normalized product of the spectral energy and frequency, $f \times \frac{E(f)}{f_s}$. This term represents energy content whereas the $E(f)$ term represents only the amplitude of fluctuation. This quantity is divided by the sampling frequency, $f_s$, to retain the units of turbulence kinetic energy (m$^2$.s$^{-2}$). In
this way, the energy of an eddy occurring at a specific frequency is plotted with respect to that frequency on the X-axis. These renormalized plots for the two span locations are shown in fig. 7.5.11 and 7.5.12. They show that eddies having frequencies above 10 Hz (note the log scale on the X axis) have a significant amount of energy. Most of the distributions for the different points on plane 3 are distributed with a similar shape, as the spectra for plane 3 varied the most for frequencies below 10 Hz, which carry low energies.

7.5.3. Rate of Turbulence Dissipation

The rate of turbulence dissipation, $\varepsilon$, is important to evaluate as it is used to calculate most of the turbulence length scales. As the turbulence is in a state of non-equilibrium, the dissipation rate is not the same for different eddy sizes. Specifically, as discussed earlier, the larger eddies are losing energy content faster than the smaller eddies. Hence, the dissipation calculated in this section is applicable for the inertial subrange and dissipation range eddies only. As the RANS models do not explicitly account for non-equilibrium turbulence, they calculate only a single value of dissipation at every node. This value is compared with the experimental dissipation rates calculated using equation 7.22. Although the experimental and computational values do not exactly represent dissipation in
the same manner, comparisons between their trends and orders of magnitude are still helpful. The inertial subrange region is identified for each plane based on where a \(-5/3^\text{rd}\) slope is observed on the power spectra. Table 7.5.1 shows the frequencies at which the dissipation was calculated for each plane. Equation 7.22 is used to evaluate the dissipation magnitudes, which are also listed in Table 7.5.1. As the previous subsection shows, the spectra at different pitchwise locations do not change much in the inertial subrange region and dissipation is evaluated at only one location per plane and is representative of that plane.

Table 7.5.1 Dissipation rates at all planes

<table>
<thead>
<tr>
<th>Plane</th>
<th>Frequency used for calculating $\varepsilon$</th>
<th>$\varepsilon$ $\text{Hz}$</th>
<th>$\text{m}^2\text{s}^{-3}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Approach</td>
<td>160</td>
<td>41.42</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>570</td>
<td>32.18</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>720</td>
<td>30.25</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>680</td>
<td>9.52</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>680</td>
<td>8.97</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>720</td>
<td>12.57</td>
<td></td>
</tr>
</tbody>
</table>

Figure 7.5.11 Pitchwise Variation in re-normalized spectral distribution for plane 3 at $Z/S_{local} = 0.5$

Figure 7.5.12 Pitchwise Variation in re-normalized spectral distribution for plane 3 at $Z/S_{local} = 0.664$
Figure 7.5.13 shows the experimental and computed turbulence dissipation throughout the test section. For experiments, the dissipation measured at the approach plane was 41.42 m².s⁻³ and is found to be the maximum of the test section. Downstream of the approach plane, the dissipation decreases monotonically as the throat (X/Cax = 0.776) is approached. It increases slightly from the throat to the downstream end of the passage. As the net turbulence kinetic energy is decreased through the passage, there is less and less energy left to dissipate, leading to a decrease in magnitude of the dissipation rate. On the other hand, the computational models show an increase in turbulence dissipation throughout the passage (0<X/Cax<1). Although incorrect, these results are in line with the computed values of the turbulence kinetic energy (see the plot of RMS velocity fluctuations as the energy and fluctuations are related to each other, as shown in equation 7.26). As the energy by both models was computed to increase in the passage, dissipation also increased. The computations seem to have failed in predicting both the turbulence kinetic energy content and the rate at which this energy is dissipated in the vane passage. This shows that the RANS models may not be suitable for computing vane passage flows and, by extension, flows that can have significant anisotropic characteristics. In the case of this passage flow.
study, the anisotropy comes about due to streamwise acceleration and deceleration, and possibly also curvature.

7.5.4. Length Scales of Turbulence

The power spectral distributions showed different trends at different vane passage locations. This was a result of redistribution of the turbulence kinetic energy among the different eddy sizes leading to non-equilibrium turbulence in the accelerating flow. This implies that the turbulence length scales will also change as the flow moves downstream. Hence, using the definitions reported in section 7.3, various length scales were calculated from the measurements. Note that the expressions given in that section were developed for isotropic, equilibrium turbulence. In the vane passage, the flow is expected to be anisotropic with non-equilibrium turbulence. Secondly, calculation of dissipation, $\varepsilon$, using equation 7.23 has some uncertainty as the spectra are not exactly straight lines. Hence, the magnitudes found from the length scale expressions are probably only approximate. Nonetheless, the actual trends for different length scales are expected to be like those found using these expressions and are expected to shed some light on the turbulence eddy sizes.

**Integral Length Scale**

The integral length scale is a characteristic of the larger turbulent eddies in the flow. They are influenced by flow geometry and, hence, different planes are expected to have different integral length scale magnitudes. Even on a given plane, the magnitudes can vary significantly. Hence, pitchwise variations in the integral length scales are documented for all planes. To understand how an integral length scale changes over a plane, it is important to understand how the parameters that are in its expression change (see equation 7.12, which is repeated below for ease of reading). These are (i) the spectral energy at the lowest measured frequency, (ii) local RMS velocity fluctuations and, (iii) local mean velocity. Of these, the first two are chosen for plotting alongside the integral length scale, as they are features of turbulence. Finally, an integral time scale is defined as follows:
The integral time scale is not dependent on the local mean velocity, which makes it dependent upon only the turbulence features. Spanwise-averaged values for each quantity are calculated, resulting in five plot points corresponding to five pitch locations. This is done because the variation in the spanwise direction is much smaller than the variation in the pitchwise direction. Still, to give the reader a sense of variability in the spanwise direction, the maximum and minimum values corresponding to each pitch location are also plotted in the same plots. The red data points correspond to maxima while the blue data points correspond to minima.
The extreme values for each pitch location also show that the trends seen for the average values (black plotlines) are not an artifact of the scatter in measurements but are due to flow physics.

Four quantities for each plane are plotted: RMS velocity fluctuations, turbulence kinetic energy (TKE) at the lowest frequency, integral length scale, and integral time scale. Figure 7.5.14 shows the plots for plane 1. The TKE plot shows large variations from pressure surface to suction surface. There is no clear trend as the scatter for each average value is high (see the accompanying red and blue data points). The RMS velocity fluctuations seem relatively unchanged. Despite changing TKE, both integral scales do not change appreciably. Equation 7.12 shows that this can only mean that the mean velocity, $\bar{U}$, decreases as the TKE...
increases, resulting in only slight changes in length scales. For plane 2, observations like plane 1 can be made (see fig. 7.5.15). The only difference is that the TKE plots show a definite decrease in energy near the suction surface and show overall less scatter compared to plane 1. At plane 2, the flow has passed over the convex (suction) surface near pitch locations 4 and 5. The acceleration caused by curvature can lead to stretching of larger eddies which dissipate more easily than the smaller eddies, leading to lower values for TKE in the smaller-eddy range.

Figure 7.5.16 Plane 3: Pitchwise variation in parameters influencing integral scales (a) TKE at Lowest Frequency, (b) RMS Velocity Fluctuations, (c) Integral Length Scale, (d) Integral Time Scale
The plane 3 plots are shown in fig. 7.5.16. Near the suction surface, the flow is like the flow over the downstream half of a sphere. This means that the flow will have some unsteadiness which can give rise to some velocity fluctuations. This is observed in the TKE plots where the energy increases from the pressure surface to suction surface, which is exactly opposite to the trend seen on plane 2. Like the upstream planes, the RMS velocity fluctuations do not change over plane 3. As both the mean velocity and TKE increase from the pressure surface to the suction surface, the integral length scales also increase (see equation 7.12). The TKE increase leads also to an increasing trend for integral time scales (see equation 7.28). Figure 7.5.17 shows the plots for plane 4. The mean velocity at this plane remains approximately constant. The TKE variation, although present, does not

Figure 7.5.17 Plane 4: Pitchwise variation in parameters influencing integral scales (a) TKE at Lowest Frequency, (b) RMS Velocity Fluctuations, (c) Integral Length Scale, (d) Integral Time Scale Note that there are only three pitch locations measured on this plane.
show a trend. Still, the TKE does not change much. The RMS velocity fluctuations do show a decrease from pressure surface to suction surface. The integral scales generally show an increase from the pressure surface to suction surface due to slight increases in TKE. Plots for plane 5, the passage exit plane, are shown in fig. 7.5.18. Compared to upstream planes, all quantities vary very little on this plane. This is expected as the exit flow is supposed to have a flat velocity profile and no variation in turbulence features along the plane is either expected or desired.

The axial variation in the integral length scale is plotted in fig. 7.5.19. The integral length scale decreases slightly from the approach plane to the inlet plane. It then rises slightly up to plane 3 (X/Cax =0.289). Downstream, it experiences a rapid increase in magnitude. Based on the expression for integral length scale
(equation 7.12) and the parameters that it depends upon, it can be concluded that this high increase is due to (i) an increase in mean velocity, \( \bar{v} \), (ii) a decrease in RMS velocity fluctuations, \( u' \), and (iii) no appreciable changes in the turbulence energy at low frequencies. As the flow is highly anisotropic in the downstream part of the passage and because the integral length scales are calculated for the streamwise velocity component only, there are too little data to explain this trend.

**Remaining Turbulence Scales**

The energy length scale and the Taylor microscale depend on two flow properties: the dissipation rate and RMS velocity fluctuations. The Kolmogorov scale depends on only one flow property, the dissipation rate. It has been shown previously in subsection 7.5.2 that the spectral distribution in the pitchwise direction does not change significantly in the inertial subrange region. As this region of the spectrum is used to calculate dissipation rate, the magnitude of the dissipation on any single plane is not expected to change much. Additionally, the RMS velocity fluctuations also do not change on a given plane (see fig. 7.5.2). Hence, the turbulence scales discussed in this subsection are expected to change only in the axial direction and an analysis of pitchwise variation of these length scales on each plane (like the one presented for the integral length scale) is not performed.
Axial Variation in Turbulence Length Scales

This subsection discusses Taylor microscales, Kolmogorov scales, and energy scales throughout the passage. The experimental and computational Taylor microscales are shown in fig. 7.5.20. Experimental results show a decrease in the magnitude up to X/Cax = 0.289, where there is very mild acceleration. Downstream, the length scale increases up to the throat and decreases further downstream as the flow decelerates from the throat to the exit of the passage. This trend is opposite to the trend of turbulence dissipation rate as higher dissipation leads to a lower Taylor microscale, as seen in equation 7.16. As a similar relation is present between the dissipation rate and Kolmogorov scale, this scale shows trends like those of the Taylor microscale as seen in fig. 7.5.21. As the computations did not correctly predict dissipation rates, it is expected that the computed length scales do not match the experimental measurements. This can be seen for both the Taylor microscale and Kolmogorov length scale plots. The computational trends are approximately opposite of the trends of the experiments.

As the energy length scale does not have a good physical significance, it is difficult to comment on its trends. Generally, the energy length scale decreases with decreasing turbulence level and increases when the turbulence level rises. As the energy length scale is a function of the turbulence kinetic energy and the
turbulence dissipation, the above observation seems logical. The computations did not correctly predict the turbulence kinetic energy and turbulence dissipation in the vane passage. As the energy length scales are evaluated based on these two quantities, the equivalent length scales calculated for the computations were not expected to match the experimental length scales. Figures 7.5.22 shows the comparison for energy length scales.

7.6. Conclusions

The presented study made a comparison between the experimental and computational turbulence characteristics of a flow through a vane passage. The experiments were conducted in a wind tunnel having a three-vane, two-passage cascade using hot-wire anemometry. The computations were performed using RANS modeling with the k-ω standard and k-ω SST turbulence closure models. Velocity fluctuations, i.e., the turbulence levels, throughout the passage were not correctly predicted in the computations. The turbulence intensity trends compared well between the experiments and the simulations, but this took place only because the intensities are calculated when turbulence levels are normalized by the mean velocities, which influenced the intensity magnitudes significantly. The RANS models could not correctly predict the turbulence kinetic energy change throughout the passage. The inaccuracy was thought to be due to the assumption of isotropic turbulence in these models. This assumption is not valid for flows with high acceleration or curvature and leads to incorrect results, as demonstrated herein. The power spectral distribution based on the streamwise component of velocity was calculated from the measurements and was found to change significantly under the influence of acceleration. Specifically, the larger eddies were found to lose their energy content more strongly with downstream distance due to stretching by flow acceleration. This leads to higher energy content in the inertial subrange of the spectra compared to spectra from a non-accelerated flow. For any axial location inside the passage, the turbulence length scales change with pitchwise location. However, the RMS velocity fluctuations remained relatively uniform anywhere on a particular plane. Finally, the RANS models do not properly
predict the dissipation rate of turbulence. This creates a huge discrepancy between measured and computed turbulence length scales. The results of this study show that RANS models are not suitable for accurate prediction of vane passage flows. Instead, Large Eddy Simulations (LES) or Detached Eddy Simulations (DES) should be considered for possible improvement, even though they may be more difficult to set up and are expected to take longer computational times.
Chapter 8. Concluding Remarks

8.1. Summary

This thesis has presented three experimental studies in the field of gas turbine passage flow and film cooling of vane passage surfaces. The general theme of this research is to present ways by which the coolant requirement in the first stage of gas turbine vanes can be decreased. A variety of measurements were taken in a wind tunnel that represents the exit of an engine combustor and the first stage of gas turbine vanes to achieve this goal. Aerodynamic measurements were taken to document the secondary flowfield features. Transport of coolant was recorded by taking coolant concentration measurements. High-frequency velocity measurements were collected to characterize the turbulence features of the passage flow. These careful measurements have been sufficient to describe the vane passage flow physics.

The first study recorded the performance of a new coolant injection scheme that utilized the coolant introduced to cool the combustor liner wall to cool the endwall and vane surfaces. This injection scheme was tested for several coolant flowrates to understand how the passage flow physics and vane passage surface cooling effectiveness were affected by changes in coolant flowrate. It was found that the new design of combustor-turbine interface, called the ‘close-coupled’ interface provided a better overall cooling effectiveness than the ‘engine-representative’ combustor-turbine interface on the endwall and the pressure surface. The lack of gap between the combustor and turbine stages, the proximity of the impingement vortex to the endwall-pressure-surface corner, and the choice of coolant flowrates were found to be the factors that increased the performance of the coolant injection scheme of the close-coupled interface. The cooling performance on the suction surface was not improved by the new coolant injection scheme due to lack of coolant migration toward the suction surface. This information is helpful to gas turbine designers to modify existing coolant injection schemes.
The presence of combustor coolant in the vane passage significantly influences the secondary flows generated in the passage. However, it was suspected that depending on combustor coolant flowrate, the generated secondary flows can either belong to the 'passage-vortex secondary flow system' or the 'impingement vortex secondary flow system.' Hence, the second study in this thesis consisted of taking aerodynamic measurements over a wide range of combustor coolant flowrates to understand how one secondary flow system evolves into another secondary flow system. When there was no coolant injection or if the coolant flowrate was low (up to a coolant MFR of 2.6%), the passage vortex system was formed. Between the coolant MFR values of 2.6% and 4.6%, neither of the secondary flow systems showed dominance. For coolant MFR values above 4.6%, the impingement vortex system was observed. One of the main features for the shift from one system to the other was the ratio of the velocity near the endwall to the velocity in the mainstream at any particular pitch location. Velocity ratios lower than unity resulted in the formation of an impingement vortex while the ratios higher than unity formed the passage vortex. The acceleration of near-endwall flow due to contouring of the endwall aided formation of the impingement vortex.

The final study in this thesis involved understanding the decay of turbulence in the vane passage. As the mainstream turbulence changes the film cooling performance of the passage surfaces, analyzing this decay is important. In industry, Reynolds-Averaged Navier-Stokes (RANS) turbulence closure models are used to compute gas turbine flows. Therefore, a RANS simulation of the wind tunnel facility was performed to compare the calculated and measured turbulence features. The RANS models did not predict the turbulence decay correctly as the flow in the wind tunnel is anisotropic while the models use the assumption of isotropic turbulence. Detailed documentation of the different turbulence length scales throughout the vane passage was provided to help gas turbine modelers to improve the accuracy of turbulence closure models.
8.2. Future Work

Several avenues can be pursued using the results presented in this thesis that can further enhance film cooling performance in the vane passage. Two of such possible future studies are outlined below.

The close-coupled interface did not improve the performance of the suction surface phantom cooling. This can be remedied by artificially increasing the amount of coolant flow injected near the suction surface compared to the coolant flowrate injected near the pressure surface. The most practical way of achieving this pitchwise skewing coolant flowrate is to enlarge the film cooling holes on the endwall near the suction surface. The increased area of the holes will reduce flow resistance leading to more coolant injection through these holes. This additional coolant is expected to help in retaining more coolant near the suction surface side of the endwall and provide phantom cooling to the suction surface. Aerodynamic and thermal measurements similar to those presented in this study can be recorded and surface cooling performance of the uniform injection scheme (presented in this thesis) and the skewed injection scheme can be compared. For the proposed study, the increase in the suction surface phantom cooling will likely come at the cost of reducing coolant near the pressure surface (as the net coolant flowrate will be the same as with the uniform injection scheme), various coolant skewing cases must be studied to find a good balance between the cooling performances of the two vane surfaces.

Although the turbulence decay measurements were taken at a sufficient number of locations throughout the vane passage, a single-wire hot-wire probe was used to measure the velocity fluctuations. Therefore, only the streamwise components of velocity and, thus, only changes in streamwise turbulence features were recorded. This information was adequate for the study presented in this chapter as it was able to show that the RANS models did not capture the turbulence correctly due to anisotropy of the vane passage flow. Therefore, the next step in this research would be to take velocity fluctuation measurements in all three principal directions throughout the vane passage flow. A triple-wire hot-wire probe will be needed to collect these data. This will complicate both the process of
recording the measurements and their analyses. But if performed, this study is bound to provide a complete picture of the evolution of turbulence in the vane cascade. This information will be helpful not only to modelers who work on creating and improving the RANS turbulence models, but also to those who work on Large Eddy Simulation (LES) models, which are known to provide more accurate solutions to complex turbulent flows.
References


[74] “TSI Thermal Anemometry Probes Catalog,” TSI Inc.


Appendix A. Uncertainty Analysis

The following section is a verbatim copy of the appendix in the master's thesis of the author (Nawathe (2019) [a]). Although the uncertainty analysis was performed again for the studies presented in this thesis, the process has been identical. This section is reproduced here only to provide an easier verification to the uncertainty magnitudes reported in this thesis. In some places, the uncertainty analysis yielded slightly different results than the ones presented in Nawathe (2019) [a]. The values in this section are updated to reflect these new results.

The calculation of uncertainties of the results presented in Chapters 5 through 7 is shown in this appendix. Instrumentation information can be found in Chapter 3. It is not possible to calculate the uncertainty for every measured location in the vane passage due to the volume of data recorded, but as the parameters being measured have different values, the uncertainties for all measured locations are not the same. However, the range of measured parameters is not very large and, thus, an uncertainty measured at a random location can be said to be approximately equal to the mean uncertainty of any location. Additionally, as seen in Chapters 5 through 7, the analysis performed in this report is more qualitative in nature than quantitative and a very high accuracy of the measured quantities was not essential in drawing conclusions. Still, the uncertainty calculations are presented to accompany the experimental results.

The Monte Carlo method (MCM) for propagation of uncertainties was used to evaluate uncertainties in measurements. A brief explanation of the method is given here, which is extracted from Coleman and Steele (2009) [b]. For each of the parameters that can bring uncertainty to the experiment, measured values (called nominal values) are first collected. Then, a standard deviation for each parameter is either decided upon (based on observations) or assumed. Then, a probability distribution function for every parameter is chosen (a normal distribution was chosen for all calculations in this study), and the distribution plot is generated. Then, a random sample of each parameter is chosen from these distributions and the final result is calculated. This process is repeated for a large number of times.
(5000 iterations for this study) and then the results of all these iterations are compiled. This compilation can now be used to form a probability distribution of the results (again, normal distribution for this study) and the standard deviation and mean of the final results can be extracted. The uncertainty calculation procedure for each of the major results is explained now:

A.1. Endwall Effectiveness and Temperature Recovery Coefficient

Thermocouples were used to measure the temperatures, which were normalized in either the endwall effectiveness ($\eta$) or the recovery temperature coefficient ($\theta$). The uncertainty was calculated based on (i) the uncertainty in measuring the temperature by the data acquisition unit, Agilent 34970A, (ii) the uncertainty in the calibration setup, (iii) the curve-fit error in the curve generated by the calibration to evaluate temperatures, and (iv) the error in not maintaining effusion coolant and film coolant temperatures to be the same. The standard deviations for all these parameters are tabulated in table A.1.1.

<table>
<thead>
<tr>
<th>Error parameter</th>
<th>Standard deviation</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Data acquisition unit</td>
<td>0.0167 °C</td>
<td>Given in equipment user guide [c]</td>
</tr>
<tr>
<td>Calibration setup</td>
<td>0.02 °C</td>
<td>Assumed based on the setup</td>
</tr>
<tr>
<td>Calibration curve-fit error</td>
<td>-</td>
<td>Integrated in the final result implicitly</td>
</tr>
<tr>
<td>Coolant temperature mismatch</td>
<td>0.067 °C</td>
<td>Constrained during measurements</td>
</tr>
</tbody>
</table>

Based on the standard deviations, a normal distribution was plotted for each parameter and the results were calculated. The data used for these calculations were taken on Plane 3 in the endwall-suction surface region, chosen at random. The mean and standard deviation for the temperature recovery coefficient, $\theta$, were 0.7256 and 0.0058. Therefore, the result for this location would be $0.7256 \pm 0.0116$. This gives an uncertainty of $\pm 1.6\%$ within the 95% confidence interval. As
the same instrument is used to measure the surface cooling effectiveness, the uncertainty in the values of $\eta$ is the same as the uncertainty in the values of $\theta$.

**A.2. Velocities Measured by Five-hole Probe**

A five-hole probe was used to measure the velocity components of the passage flow. Three types of results were used to plot the aerodynamic measurements: (i) absolute velocity of the flow, (ii) magnitudes of the velocity vectors projected onto the pitch-span (y-z) plane and (iii) angles of the vectors mentioned in (ii) relative to the span (z) axis. Result types (ii) and (iii) can be seen in the vorticity contour plots and a representative vector is plotted in fig. A.2.1. The velocity vector magnitude, $V_S$ and the vector angle, $\beta$, are defined as follows:

\[
V_S = \sqrt{\cos^2(\gamma) \cdot \sin^2(\alpha) + \sin^2(\gamma)}
\]  
\[
\beta = \tan^{-1}\{\cot(\gamma) \cdot \sin(\alpha)\}
\]

Where $V$ is the absolute velocity of the flow, $\alpha$ is the pitch angle and $\gamma$ is the yaw angle.

The uncertainties involved in calculation of flow components are as follows: (i) the error in measuring the voltage by the data acquisition unit, Agilent 34970A, (ii) the error in curve-fitting the measured voltage into pressure measured by a transducer, (iii) the error in the calibration setup of the transducer, and (iv) the error in the contour-fit interpolation used for calculating the pressure coefficients. The

![Figure A.2.1](image_url)  
*Figure A.2.1* A typical projection of velocity vector seen in plotted results. The length of the vector is equal to the magnitude of the velocity and $\beta$ is the angle the vector makes with z-axis.
standard deviations are reported in table A.2.1. The data used for calculations were chosen randomly from the mid-span, mid-pitch region of plane 6.

Table A.2.1 Uncertain parameters in the in-field velocity measurements

<table>
<thead>
<tr>
<th>Error parameter</th>
<th>Standard deviation</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Data acquisition unit</td>
<td>20 µV</td>
<td>Given in equipment user guide [c]</td>
</tr>
<tr>
<td>Curve-fit error for transducer</td>
<td>-</td>
<td>Integrated in the final result implicitly</td>
</tr>
<tr>
<td>Transducer calibration setup</td>
<td>8 Pa</td>
<td>Assumed based on the manometer used for calibration</td>
</tr>
<tr>
<td>Contour-fit error for five-hole probe</td>
<td>(a) 0.5 degree for pitch and yaw</td>
<td>Assumed after testing random locations on the calibration setup and comparing measured values with values calculated from the contour</td>
</tr>
<tr>
<td></td>
<td>(b) 0.02 for Cp</td>
<td></td>
</tr>
</tbody>
</table>

**Uncertainty in Absolute Velocity**

The mean velocity was measured to be 68.94 m.s\(^{-1}\) with a standard deviation of 0.862 m.s\(^{-1}\). This makes the uncertainty in the absolute velocity, when measured with a five-hole probe, to be ±2.5% within the 95% confidence interval.

**Uncertainty in Flow Vectors Projection**

This uncertainty can be reported in two parts: an uncertainty in the magnitude of the flow vector and an uncertainty in the angle of the vector with the z-axis. The mean of the flow vector magnitude was found to be 11.96 m.s\(^{-1}\) with a standard deviation of 0.349 m.s\(^{-1}\). This makes the uncertainty ±5.8% within the 95% confidence interval. The mean flow vector angle, β, was calculated to be -29.6° while the standard deviation was 1.55°. This gives a rather large uncertainty of ±3.1°. But it should be noted that this angle is calculated using both pitch and yaw angles. If the uncertainty of only the pitch or yaw angle is considered, it is about ±2°, which is comparable to the uncertainties reported for flow angles in the literature.
A.3. Velocities Measured by Hot-wire Anemometer

Hot-wire anemometry was used to take the approach flow velocity measurements to characterize the flow leaving the combustor. The calibration setup was from an experiment by Wilson [d] (discussed in section 4.3.2) and the uncertainty for the setup was taken from his thesis. Therefore, the hot-wire uncertainty calculation involves uncertainties due to (i) voltage measurement by the data acquisition unit (ii) calibration setup and (iii) calibration curve fit. The hot-wire calibration equation used during measurements is shown below:

\[ u^{0.3704} = 1.092 \times V^2 - 0.4993 \]  \hspace{1cm} (A.3)

Where \( u \) is the flow velocity (m.s\(^{-1}\)) and \( V \) is the measured voltage (V).

<table>
<thead>
<tr>
<th>Error parameter</th>
<th>Standard deviation</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Data acquisition unit</td>
<td>20 µV</td>
<td>Given in equipment user guide [c]</td>
</tr>
<tr>
<td>Calibration setup</td>
<td>4.6 mV</td>
<td>Given by Wilson [d]</td>
</tr>
<tr>
<td>Hot-wire calibration curve fit</td>
<td>-</td>
<td>Integrated in the final result implicitly</td>
</tr>
</tbody>
</table>

The uncertain parameters for the hot-wire measurements are compiled in table A.3.1. The mean velocity was 13.829 m.s\(^{-1}\) and the standard deviation was 0.264 m.s\(^{-1}\). The uncertainty based on this standard deviation was calculated to be ±3.8% within the 95% confidence interval.
The uncertainties for all major reported results are compiled in table A.3.2:

Table A.3.2 The uncertainties of various measured parameters

<table>
<thead>
<tr>
<th>Instrument</th>
<th>Quantity</th>
<th>Unit</th>
<th>Mean</th>
<th>Standard deviation</th>
<th>Uncertainty (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thermocouple</td>
<td>θ and η</td>
<td>-</td>
<td>0.726</td>
<td>0.0058</td>
<td>1.6</td>
</tr>
<tr>
<td>Five-hole Probe</td>
<td>Absolute velocity</td>
<td>m.s⁻¹</td>
<td>68.94</td>
<td>0.86</td>
<td>2.5</td>
</tr>
<tr>
<td></td>
<td>Projected flow</td>
<td>m.s⁻¹</td>
<td>11.96</td>
<td>0.35</td>
<td>5.8</td>
</tr>
<tr>
<td></td>
<td>magnitude</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Projected flow</td>
<td>°</td>
<td>-29.6</td>
<td>1.55</td>
<td>10.5</td>
</tr>
<tr>
<td></td>
<td>angle</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Hot-wire probe</td>
<td>Absolute velocity</td>
<td>m.s⁻¹</td>
<td>13.83</td>
<td>0.26</td>
<td>3.8</td>
</tr>
</tbody>
</table>

Note once again that these uncertainties were evaluated at a randomly chosen location (a different location for each instrument) and therefore they will not be exactly equal for all locations in the passage. Nonetheless, they give a good idea about the uncertainty of the plots presented in the results sections. As most of the conclusions drawn do not use the exact quantities but rather address general trends, the uncertainties in these experiments are suitable.

References


[c] Agilent 34970A/34972A Data Acquisition / Switch Unit User’s Guide