RQL COMBUSTOR DILUTION HOLE PLACEMENT
AND ITS EFFECT ON THE TURBINE INLET FLOWFIELD

A Thesis in
Mechanical Engineering
by
Michael J. Leonetti

©2016 Michael J. Leonetti

Submitted in Partial Fulfillment
of the Requirements
for the Degree of

Master of Science

May 2016
The thesis of Michael J. Leonetti was reviewed and approved* by the following:

Stephen P. Lynch
Assistant Professor of Mechanical Engineering
Thesis Co-Adviser

Jacqueline O’Connor
Assistant Professor of Mechanical Engineering
Thesis Co-Adviser

Mary Frecker
Professor of Mechanical Engineering
Professor-In-Charge of MNE Graduate Programs

Savas Yavuzkurt
Professor of Mechanical Engineering

*Signatures are on file in the Graduate School
ABSTRACT

RQL COMBUSTOR DILUTION HOLE PLACEMENT
AND ITS EFFECT ON THE TURBINE INLET FLOWFIELD

Dilution jets in a gas turbine combustor are used to oxidize remaining fuel from the main flame zone in the combustor, and to homogenize the temperature field upstream of the turbine section through highly turbulent mixing. The high-momentum injection generates high levels of turbulence and very effective turbulent mixing. However, mean flow distortions and large-scale turbulence can persist into the turbine section. Traditionally, combustor exit flowfield profiles have been obtained without the presence of vanes, and turbine inlet conditions have generally been simulated by the use of artificial turbulence generators (bar grids). Little work has been done to capture the details of the impact of the combustor’s turbulent flowfield on the turbine vane in a combined configuration. In this study, a dilution hole configuration was scaled from a RQL combustor and used in conjunction with a linear vane cascade in a large-scale, low-speed wind tunnel. Mean and turbulent flowfield data were obtained at the vane leading edge with the use of high-speed particle image velocimetry to help quantify the effect of the dilution jets in the turbine section. The dilution hole pattern was shifted (clocked) for two positions such that a large dilution jet was located directly upstream of a vane, or in-between vanes. Time-averaged results show that the large dilution jets have a significant impact on the magnitude and orientation of the flow entering the turbine. Turbulence levels of 40% or greater were observed approaching the vane leading edge, with integral length scales of approximately 40% of the dilution jet diameter. Incidence angle, turbulence levels, and localized pockets of high-velocity regions were dependent on the position of the dilution jets relative to the vane.
# TABLE OF CONTENTS

List of Figures .............................................................................................................. v

List of Tables ................................................................................................................ vii

Nomenclature ................................................................................................................ viii

Acknowledgements ...................................................................................................... x

Chapter 1  Introduction ................................................................................................. 1
  1.1 Relevant Literature.................................................................................................. 2

Chapter 2  Experimental Setup ....................................................................................... 6
  2.1 Uncertainty Analysis .............................................................................................. 11
  2.2 Benchmarking ....................................................................................................... 12

Chapter 3.  Results ........................................................................................................... 19
  3.1 Time-averaged Flow Structures .......................................................................... 19
  3.2 Turbulence Levels and Integral Length Scale ..................................................... 25

Chapter 4.  Conclusion .................................................................................................. 35

References .................................................................................................................... 37
LIST OF FIGURES

Figure 1. Large-scale low-speed wind tunnel facility. ............................................................... 7

Figure 2. Sketches of (a) incidence angle orientation and spanwise plane coordinate system and (b) dilution jet injection and vane leading edge location. Dotted line in (b) indicates PIV measurement plane. .................................................................................. 8

Figure 3. The two dilution hole clockings investigated: (a) Configuration 2 and (b) Configuration 1. Solid circles are OD dilution holes, while dashed circles represent the opposing ID holes. .............................................. 9

Figure 4. Measured pressure loading for the linear cascade used in this experiment (markers) and the predicted vane pressure loading from a steady RANS simulation, with no dilution. .......................... 12

Figure 5. Comparison of (a) predicted and (b) measured time-average flowfield at the vane leading edge. ................................................................................................................................. 14

Figure 6. Time-averaged normalized velocity magnitude contours for three separate PIV data sets: (a) Data set 1, (b) Data set 2, and (c) Data set 3 ................................................................. 16

Figure 7. Result for the axial component of RMS velocity when using: (a) 1 data set, (b) 2 data sets, and (c) 3 data sets of 1000 images each .............................................................................. 18

Figure 8. Contours of $U_m$ through the centerline of the dilution jets in Configuration 1, taken from a computational simulation (unpublished). ............................................................................. 19

Figure 9. Normalized time-averaged velocity magnitude contours with streamlines for the (a) I=0 case, (b) Configuration 2, and (c) Configuration 1. Data is obtained at mid-span plane at the leading edge of Vane 3 .............................................................................................................................. 22

Figure 10. Time-averaged inlet flow angle across the measurement window at x/pitch=0.3 upstream of the vane, for both dilution hole clockings. ...................................................................................... 23

Figure 11. Temporal variation of the pitchwise-averaged inlet flow angle at x/pitch=0.3 upstream of the vane, for the no-dilution case and both dilution hole clockings. ...................................................... 24
Figure 12. Contours of axial turbulence levels, with time-averaged streamlines overlaid for the (a) I=0 case, (b) Configuration 2, and (c) Configuration 1 at the Vane 3 mid-span leading edge plane. ..........................27

Figure 13. Contours of axial turbulence intensity normalized by local velocity, with time-averaged streamlines overlaid for (a) Configuration 2 and (b) Configuration 1. Data is obtained at mid-span plane at the leading edge of Vane 3..................................................................................................................................28

Figure 14. Contours of w-component turbulence levels with time-averaged streamlines overlaid for the (a) I=0 case, (b) Configuration 2, and (c) Configuration 1. Data is obtained at mid-span plane at the leading edge of Vane 3. ...........................................................................................................................................29

Figure 15. Turbulence levels across the measurement plane located at x/pitch=0.3 upstream of the vane at mid-span..................................................................................................................................................................................30

Figure 16. Example of (a) a snapshot of axial velocity fluctuation (m/s) and (b) the temporal autocorrelation of axial velocity at the white circle in the contour plot for Configuration 1 and I=32.7. Data is obtained at mid-span plane at the leading edge of Vane 3. .................................................................32

Figure 17. Example of (a) axial velocity fluctuation (m/s) and (b) the spatial autocorrelation of axial velocity (including an exponential fit) along the white line in the contour plot for the Configuration 1 and I=32.7. Data is obtained at mid-span plane at the leading edge of Vane 3..................................................33

Figure 18. Comparisons of the axial integral length scale at x/pitch=0.3 and vane mid-span using the temporal and spatial autocorrelations........................................................................................................................................34
LIST OF TABLES

Table 1. +/- Percent precision uncertainty for the three test cases at (x/p=0.3 and z/p=0) ........................................11
Table 2. +/- Percent total uncertainty for the three test cases at (x/p=0.3 and z/p=0) .............................................12
Table 3. Turbulence levels for both clockings at x/pitch=0.3 and z/pitch=0 ..........................................................30
Table 4. Average integral axial length scale at x/pitch=0.3 for the ODID4 clocking .................................................34
NOMENCLATURE

\( C_{ax} \)  
Vane axial chord

\( D \)  
Dilution jet diameter

\( I = \frac{(\rho U^2)_{jet}}{(\rho U^2)_{\infty}} \)  
Momentum flux ratio

\( ID \)  
Inner diameter panel of the scaled combustor

\( L_{x-s} = \int_0^\infty R_{ii}(x, z, \Delta x) \, d\Delta x \)  
Spatial axial integral length scale

\( L_{x-t} = U_m \star \int_0^\infty R_{ii}(x, z, \Delta t) \, d\Delta t \)  
Temporal axial integral length scale

\( \frac{Nu}{Nu_0} = 1 + 0.04 \times TuRe^{12} \left( \frac{L}{D} \right)^{-1/3} \)  
Ames' Correlation for stagnation point heat transfer augmentation factor

\( OD \)  
Outer diameter panel of the scaled combustor

\( P \)  
Pressure

\( R_{ii}(x, z, \Delta t) = \frac{\langle u'(x, z, t) \ast u'(x, z, t + \Delta t) \rangle}{u_{rms}(x, z)} \)  
Temporal autocorrelation coefficient

\( R_{ii}(x, z, \Delta x) = \frac{\langle u'(x, z, t) \ast u'(x + \Delta x, z, t) \rangle}{u_{rms}(x, z) \ast u_{rms}(x + \Delta x, z)} \)  
Spatial autocorrelation coefficient

\( S \)  
Test-section span

\( Tu_{loc} = \frac{u_{rms}}{U_{loc}} \)  
Axial component of turbulence intensity normalized by the local velocity magnitude

\( Tu_m = \frac{u_{rms}}{U_{m-avg}} \)  
Axial component of turbulence levels normalized by the turbine inlet mass-average velocity magnitude

\( Tw_m = \frac{w_{rms}}{U_{m-avg}} \)  
Pitchwise component of turbulence levels normalized by the turbine inlet mass-average velocity magnitude

\( U_{jet-plenum} = \sqrt{\frac{2 \times (P_{\infty} - P_{\text{plenum}})}{\rho_{\infty}}} \)  
Dilution jet velocity using plenum and mainstream pressure data

\( U_{jet-pitot} = \sqrt{\frac{2 \times \Delta P}{\rho_{\text{jet}}}} \)  
Dilution jet velocity determined using pitot probe data
\( U_m = \frac{U}{U_{m-avg}} \)  
Velocity magnitude normalized by the turbine inlet mass average velocity

\( U_{m-avg} = \frac{m_{\text{combustor}} + \sum m_{\text{dilution jets}}}{\rho_\infty * S * W} \)  
Mass-averaged turbine inlet velocity magnitude

\( W \)  
Combustor width

\( \dot{m} = \rho U A \)  
Massflow rate

\( p \)  
Vane pitch

\( u \)  
Axial component of velocity

\( v \)  
Spanwise component of velocity

\( w \)  
Pitchwise component of velocity

\( x \)  
Axial direction

\( y \)  
Spanwise direction

\( z \)  
Pitchwise direction

Greek

\( \varphi = 0.00851 \sqrt{TuRe_d^{5/12}} + 1 \)  
Van Fossen\(^9\) Correlation for stagnation point heat transfer augmentation factor

\( \alpha \)  
Incidence angle

\( \rho \)  
Density

\( \nu \)  
Kinematic viscosity

Subscripts

\( OD4 \)  
An OD4 dilution jet property

\( jet \)  
Jet property

\( loc \)  
Normalized by a local velocity magnitude

\( m \)  
Normalized by a turbine inlet mass-average velocity magnitude

\( m-avg \)  
Mass-averaged value

\( rms \)  
Root mean square

\( \infty \)  
Mainstream property
ACKNOWLEDGEMENTS

I would like to start by thanking my two advisers, Dr. Stephen Lynch and Dr. Jacqueline O’Connor. They both took me under their respective wings and helped me navigate the dangerous world of computational and (especially) experimental gas turbine research. They always made themselves available to me, answering any questions I had and giving advice that always proved useful. I’m half convinced they don’t ever sleep. They also were always glad to lend a hand in the lab when needed to illustrate a technique in person or an experimental setup. Their passion for their research showed the most during these times outside of their offices, which was sometimes late in the afternoon, and that always helped me maintain my enjoyment of the work when things would not go according to plan (Big Blue, heaters, launched dilution plates, etc.). I need to single Dr. Lynch out for bringing me in to the ExCCL lab and changing my graduate career for the better. I would like to thank United Technologies—Pratt & Whitney for their financial and technical support of this work, specifically Sean Bradshaw, who was invaluable to the research even though he was 350 miles away from Penn State.

I would like to thank all of my lab mates for their help in the lab and their friendship, as well as for pretending to laugh at my jokes. Graduate school is can be difficult, but I can’t imagine working in another lab or having as much fun (but not too much) as I did with the ExCCL lab. I would like to recognize: Robert Schroeder, for teaching me the ins-and-outs of the PIV system which was instrumental to this research; Jeffrey Gibson, for his help in modifying the Big Blue wind tunnel and my experimental test section; and Shane Haydt, for helping me lift the tunnel side glass for what seemed like an infinite number of times (as he always reminded me). I will miss working with them all. I will not, however, miss climbing inside of ole’ Big Blue.

I would also like to thank my parents and my brother for their love and support throughout my entire graduate school career, not to mention my whole life as well. Finally I would like to thank my girlfriend, Elizabeth, for her love and support, and for being my best friend; I truly could not have done this without her.

I’m very proud of the work that I was able to complete at Penn State and I have them all to thank, and for that I’ll always be grateful.
Chapter 1

Introduction

The combustor section of a gas turbine engine burns fuel and air to create the high enthalpy fluid needed to turn the turbines. In a Rich burn-Quench-Lean burn (RQL) style of combustor, popular in aviation gas turbines, large dilution jets are injected into the combustor downstream of the initial rich combustion zone to oxidize remaining fuel from the main-combustion zone. The jets also help to homogenize the flow temperature through turbulent mixing. This creates high-turbulence levels and potentially a non-uniform flowfield (if mixing is insufficient) that enters the turbine section. This oncoming flowfield can be very detrimental to the turbine vane durability since the gas temperatures can exceed the vane melting temperature. One trend in commercial aviation gas turbines is increasingly smaller engine cores to achieve ultra-high bypass ratios for high propulsive efficiency. Thus, combustors continue to shorten in length, potentially positioning dilution jets closer to the downstream vanes. While the turbine vanes are designed with advanced cooling techniques to survive the hot gas temperatures, the cooling strategy effectiveness is highly dependent on accurate knowledge of the turbine inflow conditions.

The goal of this work is to provide some understanding of the dilution jet’s impact on the flow field approaching the turbine vane through high-speed, spatially-resolved flowfield measurements. This understanding can be used to aid in the improvement of vane cooling efficiency. It has also shown the importance of integrated design between the combustor and turbine sections. The results could also be applied to improving computational predictions of turbine flow by providing temporally and spatially resolved turbulence and flowfield characteristics at the turbine inlet plane.
1.1 Relevant Literature

Many experimental and numerical studies have investigated combustor and turbine flowfields separately, but few have investigated the flowfield as it exits a combustor and enters the turbine section. For this study, we consider the dilution jets and their effect on turbulence levels and mean flow distortion. These dilution jets are effectively jets in crossflow, which have been investigated heavily in the past.

Fric & Roshko [1] showed that a jet injected into a crossflow creates four types of coherent structures in the near-field of the jet: jet shear-layer vortices, horseshoe vortices, wake vortices, and counter-rotating vortex pair. All of these structures contribute to the time-mean and turbulent flowfield downstream of the jet. However, in a gas turbine combustor, these jets are confined and in close interaction with neighboring jets. Several studies have investigated confined jet behavior in combustor-like geometries. Holdeman [2] found that jet trajectories in an annulus are similar to those in a rectangular duct for the same momentum flux ratio. Holdeman et al. [3] also found that jet penetration was dependent on momentum flux ratio, and therefore so is the flow distribution and mixing. In-line dilution jet configurations had both better initial mixing and downstream mixing for momentum flux ratios less than 64, relative to staggered dilution jet configurations. Holdeman et al. [4] compared velocity profiles of dilution jets with the same momentum flux ratio but with varying density ratios, and found that density ratio only had a second order effect on the profiles. These studies, among others, indicate that the momentum flux ratio and jet alignment have the largest effect on jet penetration and mixing.

However, many prior studies of dilution mixing have focused on time-mean results. When jets are injected into crossflow they generate a large amount of turbulence, which can impact vane heat transfer and effectiveness of cooling techniques. Most studies have found that turbulence levels entering the turbine can range between 10-20%, with integral length scales on the order of the dilution jet diameter. These results vary with dilution jet arrangement, hole size, hole location, and momentum flux ratio. Cha et al. [5] found a peak level of turbulence levels at the combustor-turbine interface of $u'/U \sim 35\%$ and length scales of up to 25% of the vane chord length. Vakil et al. [6] used a non-reacting cold-flow combustor with both dilution
and film cooling holes that had turbulence levels of 20% due to the large penetration depths of the dilution holes. Kidney shaped thermal fields were created from the counter-rotating vortices generated by the jets; this created a turbine inlet plane that had anisotropic turbulence and non-uniform thermal fields. Barringer et al. [7] conducted a similar experiment, which yielded slightly lower turbulence levels levels in the range of 15-18% for an isothermal combustor. Ames and Moffat [8] also found similar turbulence levels generated in their simulated isothermal combustor, which reached as high as 19%. This study also determined that the turbulent length scale in their flowfield was on the order of magnitude of the dilution hole diameter. These high freestream turbulence levels are well-known to augment heat transfer, particularly at the stagnation point on a vane.

Most prior studies investigating the effects of high freesteam turbulence on heat transfer augmentation have used simulated turbulence from bar grids. Van Fossen et al. [9] studied the effect of high freestream turbulence, generated by bar grids, on stagnation heat transfer. In general, stagnation region heat transfer increased with decreasing turbulence length scale and increasing freestream turbulence level. A correlation was proposed to predict the effect of augmentation of stagnation heat transfer, $\varphi$; however, this correlation was based on isotropic turbulence, which may not be appropriate for the vane in an engine. Ames [10] considered higher levels of turbulence and their effects on vane heat transfer using both bar-grid turbulence, as well as dilution jets from a simulated combustor. This study concluded that an energy scale $Lu$, which incorporates turbulent kinetic energy and length scale, had a significant impact on the stagnation and pressure surface heat transfer. Van Fossen et al. [9] used their own correlation to compare the results to an experiment with high turbulence of levels of 28% generated from a combustor. The correlation using bar grid turbulence decay ($\varphi$) underpredicted the heat transfer augmentation by 14%. Ames et al. [11] found that the grid-generated turbulence correlation underpredicted vane heat transfer augmentation, while the TRL parameter overpredicted heat transfer augmentation. The correlations of Van Fossen and Ames both bound vane stagnation heat transfer augmentation at high turbulence levels, but neither gives an accurate value, likely due to an incomplete understanding of the nature of the turbulence at high turbulence levels.
Growth in computational capability and the continued desire to optimize turbine engines has led to increased interest in simulating the combustor and turbine simultaneously so that assumptions about boundary conditions between the two are eliminated. Prior computational simulations often only focused on either the combustor or turbine section, generally keeping the two portions of the engine separate since modeling both can be computationally intensive. Experimental results have shown that the flowfield at the combustor outlet is highly-turbulent and spatially non-uniform, although often boundary conditions used at the turbine inlet do not represent this. Another potential issue from performing separate simulations is the absence of the vane’s impact in the combustor simulation; this is especially true for combustors with dilution jets positioned near the turbine inlet. Cha et al. [12] showed through experimental and computational studies that the nozzle guide vane’s (NGV) potential field has an effect on the upstream combustor flow. Cha found that the NGV’s impact occurs well before the combustor-turbine interaction plane where many combustor-only studies end. However, simply modeling the combustor and turbine together in a RANS simulation is not enough to ensure accurate results; higher fidelity unsteady models are needed to accurately predict the large turbulent structures stemming from the unsteady combustor flowfield. Salvadori et al. et al. [13] performed two RANS simulations, one where the combustor-vane interaction were modeled separately with no feedback between the two computational domains, and a second simulation with coupling between the two domains. They found that the de-coupled simulation did a poor job of predicting the flow entering the vane section. This resulted in a turbine simulation that over-predicted the impact of swirl, while also failing to capture the dilution hole clocking effects. They recommended the use of at least a coupled approach for its more accurate flowfield entering the turbine inlet. A steady RANS simulation conducted by Stitzel et al. [14] that simulated both dilution and film cooling flow showed that the use of a two-dimensional turbine inlet boundary condition was inaccurate due to the predicted combustor exit flowfield. The exit flowfield from a realistic combustor exhibited three-dimensional behaviors with non-uniformities in temperature, pressure, and velocity.
Insinna et al. [15] also used a coupled combustor/turbine simulation which found similar non-uniformities at the turbine inlet in the radial and tangential directions. Thermal differences on the vane and changes in incidence angle of the oncoming flow were also found using this coupled model. Prenter et al. [16] conducted an experiment in an annular combustor-turbine rig which included the implementation of dilution jets and reacting flow. Measurements from this experiment were then compared to that of six computational simulations each using a different steady RANS turbulence model. Each turbulence model predicted different temperature profiles at the turbine inlet plane. The RANS models also underpredicted jet mixing as well as lateral spreading. Cha et al. [17] performed both a RANS and LES simulation and compared them with experimental results, noting that the LES simulation more accurately predicted the turbulence intensities found at the combustor-turbine interface in the experiment. Cha stated that the RANS model completely misrepresented where the highest turbulent energy is produced while the LES was able to simulate most turbulent characteristics.

The purpose of this study is to experimentally characterize the mean and unsteady flowfield entering the turbine inlet to provide a more complete understanding of the incoming flow conditions. Dilution hole placement is considered in this study by alternating the pitchwise location of the holes with respect to the vanes. A dilution hole momentum flux ratio representative of RQL combustor designs is used, based on the importance of momentum flux ratio in the jet behavior as described earlier. Although heat transfer measurements were not taken during this study, the likely impact of the incoming flowfield on vane heat transfer is mentioned.
Chapter 2

Experimental Setup

This experiment utilized a simulated combustor and scaled vanes in a low-speed large-scale wind tunnel. This wind tunnel has been used in previous investigations of combustor and other dilution flow studies [6,7,14]. It is a recirculating closed-loop wind tunnel. Upstream of the test section, the flow is split into the main core flow section and two bypass flow sections. The flow can be controlled to divert the wanted amount of flow into the bypass sections. The experiments presented in this paper were conducted isothermally. The core flow was used as the mainstream combustor flow, while the two bypass flow sections were utilized as plenums to feed the dilution flow. The tunnel has the capability to insert different test sections of varying span. Vane test sections are inserted at the corner of the tunnel to complete the recirculating loop. The first vane used in the experiment was based on a commercial engine design, and is described by Gibson et al.[18] The vane test section has 5 vanes, an inlet span height of 1.912 $C_{ax}$, a pitch of 1.215 $C_{ax}$, and an inlet Reynolds number based on axial chord of 64,000. A turbulence bar grid is located upstream of the dilution to provide initial turbulent flow; without dilution it results in a 7% turbulence level at the turbine inlet. The large-scale wind tunnel does not have the capabilities to run compressible flow experiments; therefore Mach number in the cascade was not matched to engine conditions, but the Reynolds number was matched due to the large scale. Previous studies have shown that Mach number has little effect on secondary flowfields in the vane passage (Perdichizzi [19] and Hermanson [20]). Mach number was also shown to have little to no effect on pressure side heat transfer, although suction side surface pressure and heat transfer are affected by Mach number, as shown by Nealy [21] and Arts [22]. The experiment was conducted without the addition of fuel and reactive products, although studies such as those conducted by Zimmerman [23] and Moss [24] have shown that turbulence levels values downstream of combustion were unaffected by the combustion process.
For this study, a commercially-relevant RQL-style combustor geometry was scaled for the large wind tunnel and inserted upstream of the vane test section. Due to vane geometry constraints, the Holdeman [2] parameter of the scaled combustor geometry was smaller than typically expected for aeroengine combustors, which would result in some underpenetration of the dilution jets relative to an optimum configuration. Note that in this study, the combustor simulator did not have swirled flow approaching the dilution holes. The level of swirl normally present in an aeroengine RQL combustor was presumed to be negligible relative to the effect of high-momentum-flux dilution injection. Momentum flux ratio was also matched to a representative engine condition. Momentum flux ratio was determined to be the most significant aerodynamic parameter since this will determine the jet’s trajectory as discussed earlier. Since the low speed wind tunnel cannot match the density ratios found in a real engine, the mass addition of each hole was not matched to the engine condition. Note also that in the wind tunnel, the vane cascade geometry is planar (not annular), so the ID and OD walls in the wind tunnel have the same arclength. In the wind tunnel implementation, the bottom wall of the tunnel was designated as the OD endwall, and the top wall was designated as the ID endwall. This is because the direction of the vanes in the cascade is reversed relative to convention. The vanes and the coordinate system used in this experiment can be seen in Figure 2 (a & b). The dashed line in Figure 2b shows the measurement plane that was investigated in this study.
The simulated combustor consisted of two full combustor sectors, where each sector had 4 dilution holes with an alternating pattern of large and small diameter holes. The OD and ID sectors had the same number of holes. This meant that both the OD and ID sectors had 8 dilution holes each across the entire pitch of the tunnel. The dilution hole centerlines were located $1.77 C_{ax}$ upstream of the vanes, see Figure 3.

Figure 2. Sketches of (a) incidence angle orientation and spanwise plane coordinate system and (b) dilution jet injection and vane leading edge location. Dotted line in (b) indicates PIV measurement plane.

Dilution hole centerlines were directly opposed to each other, but with pitchwise staggering of hole diameters. That is, the large holes on one panel were directly opposed to small holes in the opposite panel. Figure 3 indicates the layout of the dilution holes relative to the central vane (vane 3) in the cascade. The experimental measurement plane can also be seen in this figure as noted by the dashed lines around vane 3. Note that the combustor sectors did not have effusion cooling and only consisted of a single row of dilution holes.

Two positions of the dilution holes relative to the vanes (termed “clockings”) were used during this experiment. In Configuration 1, a large dilution hole centerline was aligned to the leading edge of vane 3.
For Configuration 2, the vane stagnation line projected upstream would pass directly between the holes. The focus of this study was on the turbine inlet at the middle vane.

\[ \text{(a) Configuration 2} \]

\[ \text{(b) Configuration 1} \]

Figure 3. The two dilution hole clockings investigated: (a) Configuration 2 and (b) Configuration 1. Solid circles are OD dilution holes, while dashed circles represent the opposing ID holes.

Two values for dilution momentum flux ratio (I) were used: I=0 and I=32.7. The I=0 case was used as a benchmark to compare against the effects of no dilution. For the high momentum flux ratio of I=32.7, the trajectory of the large dilution jets was expected to impact the opposing endwalls. Each dilution panel
was fed from a separate plenum, which was fed from tunnel flow that can be diverted around the core flow region (see Figure 1).

The flowfield measurements were taken with a high-speed particle image velocimetry (PIV) system. The flow was seeded with 1 μm particles of di-ethyl hexyl sebacate which was inserted upstream of the wind tunnel fan, so that it was fully mixed into the core and dilution plenum flows. The PIV system included a Nd:YLF dual-head laser, capable of 20 mJ per pulse per head at a 1 kHz repetition rate with 170 ns pulse width. The camera used in the experiment utilized a 60mm lens and had a 1024 x 1024 pixel resolution and a capture frequency of 2,000 frames per second at full resolution. System control and synchronization was performed with LaVision software (DaVis 7). The PIV calculation was done with DaVis 8. In this study, PIV measurements were taken at a sample rate of 1000 Hz with \( dt = 30 \mu s \) between image pairs in a sample. The images were preprocessed with a particle intensity normalization to remove background intensity in the images. Geometric masks were used at the vane leading edge to remove questionable data due to laser reflections. A multi-pass method was used during PIV calculation with decreasing window size. The first two passes were done with a 64x64 window size and 50% overlap. The window size then decreased to 16x16 at 50% overlap through four passes. Minimal post-processing was completed in DaVis; poor vectors were removed from the processed images if they had a peak ratio < 1.1, which is the ratio of the correlation value of the highest and second highest peak. Over 99% of vectors used in the PIV calculations were the first choice for each data set. Very few poor vectors were found (less than 1% of all vectors), those that were found were removed and replaced with a value based on the surrounding valid vectors. An investigation of the vector statistics for each data set showed that for the Configuration 1 clocking, 99.15% of the vectors used in PIV calculation were the first choice vector. For the Configuration 2 clocking this value was slightly higher, with 99.38% of all vectors used being the first choice. The time-averaging of the vector fields was completed with the use of a MATLAB code created in-house.

The measurement location for all flowfield results discussed in this paper is found at the leading edge and midspan of vane 3. The measurement plane is the turbine inlet radial plane that captures two-
dimensional velocity \((u\text{ and } w)\) ahead of the Vane 3 leading edge as shown in Figure 3. The plane itself is \(0.61\ C_{ax}\) by \(0.61\ C_{ax}\). Two-dimensional PIV was chosen over stereoscopic PIV due to the limited optical accessibility in front of the vanes.

### 2.1 Uncertainty Analysis

Uncertainty analysis was conducted using the three data sets collected for each test condition. The full data set was split up to create a total of 6 subsets to be used in a precision uncertainty analysis. Precision uncertainty was analyzed using the method described by Moffat [25] with a 95% confidence interval. The percent uncertainty values were very low for the magnitude of velocity and both turbulent components. The length scale data did not have the same low levels of precision uncertainty. This is believed to be a result of splitting the full averaged data set into 6 subsets. When an insufficient number of images were used to calculate integral length scale, the value could fluctuate depending on the position of the dilution jets in the sample. Percent precision uncertainty is reported at a point aligned with the leading edge of the vane (the dilution clocking position) and the axial position upstream of the vanes used in data analysis seen later \((x/pitch=0.3)\), and the results are shown in Table 1.

<table>
<thead>
<tr>
<th>Case</th>
<th>Variables</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>(U)</td>
</tr>
<tr>
<td>I=0</td>
<td>0.3</td>
</tr>
<tr>
<td>Configuration 1</td>
<td>3.3</td>
</tr>
<tr>
<td>Configuration 2</td>
<td>2.8</td>
</tr>
</tbody>
</table>

The total uncertainty, which takes bias and precision uncertainty into consideration, was also calculated for the velocity magnitude quantity for each test case and is reported in Table 2. An instantaneous displacement uncertainty of +/- 0.15 pixels/pixel was estimated for the bias uncertainty in this setup. This
gives a reasonable estimate, erring on the side of caution, for the bias uncertainty calculation.[26] Note that the bias uncertainty is a significant portion of the total uncertainty in the measurements.

**Table 2. +/- Percent total uncertainty for the three test cases at (x/p=0.3 and z/p=0)**

<table>
<thead>
<tr>
<th>Case</th>
<th>U</th>
</tr>
</thead>
<tbody>
<tr>
<td>I=0</td>
<td>6.9</td>
</tr>
<tr>
<td>Configuration 1</td>
<td>10.7</td>
</tr>
<tr>
<td>Configuration 2</td>
<td>11.0</td>
</tr>
</tbody>
</table>

**2.2 Benchmarking**

Static pressure taps were located at the midspan of all five vanes to ensure that the vane test section had a periodic flowfield, without dilution flow. The experimental results were compared to results obtained from a periodic CFD simulation to ensure that the flow entering all vane passages were correct before introducing the dilution flow [18]. The vane pressure loading can be seen in Figure 4.

![Figure 4. Measured pressure loading for the linear cascade used in this experiment (markers) and the predicted vane pressure loading from a steady RANS simulation, with no dilution.](image)
Pressure measurements were taken in the plenums and in the mainstream flow upstream of the dilution holes, to estimate the average momentum flux ratio of the dilution jets. The mainstream velocity was recorded by traversing a pitot probe along both the span and pitch of the combustor upstream of the dilution jets. The dilution jet velocity was calculated using the measured freestream and plenum pressures:

\[ U_{jet-plenum} = \sqrt{\frac{2(P_{\infty} - P_{plenum})}{\rho_{\infty}}} \]  

Eq. (1)

With the measured jet and mainstream velocities, the momentum flux ratio was calculated using

\[ I = \frac{\left( \rho U^2 \right)_{jet}}{\left( \rho U^2 \right)_{\infty}} \]  

Eq. (2)

A traversable pitot probe was also used at the exit of each dilution hole to record the centerline velocities of the dilution jets as a secondary check. Dilution jet centerline velocities for individual jets were found to vary +/-6% relative to the average of all jets. The average of the centerline velocities was used to calculate a momentum flux ratio using the same equation, which was within 3% of the estimation based on Eq. (1).

An initial check was done to ensure that the PIV measurements obtained in the tunnel were accurate and properly configured, by comparing the flowfield with no dilution to a steady CFD simulation of the vane geometry. Figure 5 shows a comparison of contours of normalized velocity magnitude \( U_m \) from a steady RANS computational simulation of the vane cascade by Gibson\(^\text{19}\), with the time-average result from the PIV data taken in the measurement plane shown in Figure 3. Overlaid on the contours are streamlines. In this and subsequent figures, the velocity is normalized by the turbine inlet mass-averaged velocity \( U_{m-avg} \). This mass-average velocity is determined by measuring the velocity of the flow upstream of the dilution holes, as well as the velocity of the individual dilution jets (from measured center-line velocities). The massflow contribution of each hole is then included in the total massflow downstream of dilution injection, to determine the average turbine inlet velocity:

\[ U_{m-avg} = \frac{\dot{m}_{combustor} + \sum \dot{m}_{dilution jets}}{\rho_{\infty} S + W} \]  

Eq. (3)
Good agreement is found in Figure 5 between the CFD and PIV measurement for both magnitude of velocity, as well as for the direction of the incoming flow. Note that the region right around the vane leading edge could not be captured, due to laser reflections from the vane surface, and thus the dark blue region of invalid data at the bottom of Figure 5b is larger than the actual vane leading edge.

![Figure 5. Comparison of (a) predicted and (b) measured time-average flowfield at the vane leading edge.](image)

Another check performed was the repeatability and statistical convergence of the measurements. At least three datasets were obtained for each flow condition and dilution clocking. Due to camera memory limitations, the maximum amount of continuous samples was limited to 1000 in each dataset. Figure 6 shows a comparison of the time-average of each of the three datasets, which show good agreement.
a) $U_m$

b) $U_m$
Although the comparison above indicates reasonable sample sizes, the final results shown later use the average of all three data sets. This is done to ensure statistical stationarity in higher-order moments. The results of averaging the three data sets are shown in Figure 7. The average of three sets was deemed sufficient for stationarity in the fluctuating velocity component (presented as turbulence level in the figures).
Figure 7. Result for the axial component of RMS velocity when using: (a) 1 data set, (b) 2 data sets, and (c) 3 data sets of 1000 images each
Chapter 3.

Results

3.1 Time-averaged Flow Structures

To help orient the reader on the flowfield generated by the dilution jets upstream of the vane leading edge, Figure 8 shows a contour slice of predictions of $U_m$ from a computational simulation (unpublished) of Configuration 1, for the same momentum flux ratio as in this study. The dashed line shows the extent and location of the PIV measurement plane and the solid black line shows where the vane leading edge is located. The simulation predicts that the large dilution jet trajectory is deflected by the crossflow from left to right, but extends all the way to the upper (ID) wall and passes through the midspan upstream of the PIV plane. The corresponding small jet penetrates nearly to a quarter of the span before becoming entrained in the large jet.

![Figure 8. Contours of $U_m$ through the centerline of the dilution jets in Configuration 1, taken from a computational simulation (unpublished).](image)

Experimental measurements of the time-averaged normalized velocity magnitude are shown in Figure 9. The coordinates are normalized by the vane pitch and are set up so that $x$/pitch=0 and
z/pitch=0 correspond to the vane leading edge. For Configuration 1, the centerlines of the jets are aligned with z/pitch=0, and for Configuration 2, the vane 3 leading edge is located between holes (refer to Figure 3). As described earlier, the dark region located around x/pitch=0 and z/pitch=0 is a masked out region around the vane leading edge. This was done to exclude poor data very close to the surface of the vane due to reflections of the laser.

The top contour plot of Figure 9a shows the flow entering the turbine with no dilution flow (I=0). The overlaid streamlines show that the oncoming flow is approaching the vane at an inlet flow angle (α) of 0° until the vane pressure field begins to turn the flow around the vane. The remaining two contour plots in Figure 9 show the results for the two clockings investigated in this study, at a momentum flux of I=32.7. As described earlier, the core of the dilution jets is expected to penetrate past this plane upstream of this measurement window.

For Configuration 2 (Figure 9b), there appear to be no high-velocity remnants of the large dilution jets in this plane. It is likely that the mixing in the space between the jets has homogenized the flow fairly well. There is a larger stagnation region around the vane leading edge than is found for the I=0 case. The most striking difference between Configuration 2 and the no-dilution case is the significant change in the incoming flow direction, as indicated by the streamlines overlaid on the contours. This significant change is thought to be due to entrainment of fluid into the wake of the large OD dilution jet positioned to the left of this region (not visible in this data region), and the strong acceleration of the wake around the vane suction side.

Configuration 1 in Figure 9c shows a low-velocity region around x/pitch=0.35, z/pitch=-0.15 that is likely the wake of the large OD dilution jet directly upstream of this location. A higher velocity region (U_m=1.25) is located just to the right of it, which is part of the large ID jet that is still penetrating the span and turning in the crossflow. Although the distribution of velocity magnitude is less uniform for Configuration 1 versus Configuration 2, Figure 9 shows that the incoming flow for Configuration
1 has a less extreme angle. This measurement is nearer to the centerline of the dilution jet wake and more likely to be aligned with the average inflow direction.
Figure 9. Normalized time-averaged velocity magnitude contours with streamlines for the (a) I=0 case, (b) Configuration 2, and (c) Configuration 1. Data is obtained at mid-span plane at the leading edge of Vane 3.

Time-averaged inlet flow angles were extracted from the data set for both clockings, as well as the no dilution case for comparison, and are shown in Figure 10. The horizontal axis is the pitch direction across the measurement window (z/pitch), and the vertical axis shows the local time-averaged flow angle at a location x/pitch=0.3 upstream of the vane (x/axial-chord=0.365). The solid line in the line plot shows the flow angle for the no dilution case (I=0), which indicates that the vane’s pressure field has begun to turn the flow at this location (as expected). Configuration 1 (larger dashed-line) has a peak negative magnitude of -7.9° with an average across the pitch of the measurement window of -5.2°. This is a mild negative inlet angle, but appreciably different than the no-dilution case. Configuration 2 has a more severe negative flow angle, with a peak negative angle of -19.4° with an average of -15.2° across the measurement plane. This negative inlet angle likely has a significant impact on the location of the vane stagnation, and also might result in a small suction side separation on this airfoil, although the density of static pressure taps on the airfoil were not sufficient to determine this.
Temporal variations of the turbine inlet flow angle were also investigated, since the dilution flow is naturally unsteady. Figure 11 shows the temporal variation of the inlet flow angle, spatially averaged across the pitch at \( x/pitch=0.3 \). The mean value and standard deviation of the inlet flow angle are also indicated on the figures. The no dilution case (\( I=0 \)) shows that there is very little deviation from the mean without the presence of the unsteady dilution jets. However, dilution flow causes widely varying instantaneous inlet flow angles that can range up to +/-40°. The standard deviation for both clocking positions is almost the same, which might be expected because the unsteady turbulent breakdown of the dilution flow is similar regardless of dilution hole position. However, in a time-averaged sense, Configuration 2 results in a more negative inlet flow angle, relative to Configuration 1 upstream of the clocked vane. This is thought to be due to the low-momentum wake region behind the large OD jet being strongly accelerated toward the vane 3 suction side.
Figure 11. Temporal variation of the pitchwise-averaged inlet flow angle at x/pitch=0.3 upstream of the vane, for the no-dilution case and both dilution hole clockings.
3.2 Turbulence Levels and Integral Length Scale

RMS values of velocity were calculated from the instantaneous measurement sets for both clockings to determine turbulence levels created by the array of dilution jets. Axial turbulence levels ($ Tu_m $) values are shown in Figure 12, where only the RMS of the x-component of velocity was used. Axial velocity RMS was normalized by the mass-averaged turbine inlet velocity for each case, and not by the local velocity magnitude. The low levels of turbulence found in the no-dilution case are from the bar grid located far upstream of the dilution holes, which is expected to decay to approximately 6% at the turbine leading edge based on grid turbulence correlations [28]. For Configuration 2, the axial turbulence level entering the measurement plane was approximately 59.4%. This is much higher than the values found in literature, which generally report values in the 10-20% range. However, the reported value will certainly be a function of distance from the dilution holes and the amount of convergence of the combustor walls as the flow moves toward the vane, which are not often reported. Figure 12 also indicates that the axial turbulence level for Configuration 2 appeared to be relatively uniformly distributed across the pitch of the measurement plane, upstream of the vane. Configuration 1 also results in similar levels of high turbulence at the measurement location in front of the vane, but relative to Configuration 2, the distribution of turbulence seems less uniform, similar to the non-uniform distribution of velocity magnitude in Figure 9c.

Turbulence levels are also presented using a local velocity magnitude as the normalizing parameter, to indicate regions of very high fluctuations relative to the local flow speed which varied due to the dilution jet cores and wakes. Figure 13 indicates the turbulence levels based on a local velocity magnitude, where the turbulence levels are generally higher in regions of low-velocity flow (such as the vane stagnation; see Figure 9) and lower in high-velocity regions (around the vane suction side, on the left). For Configuration 2, there is a band of high local turbulence levels of 55% in the center of the measurement plane, where the large dilution jets are mixing with each other. However, Configuration 1 shows low local turbulence levels
toward the right (z/pitch=0.1), which are associated with the high-velocity remnant of the large ID jet described for Figure 9.
Figure 12. Contours of axial turbulence levels, with time-averaged streamlines overlaid for the (a) I=0 case, (b) Configuration 2, and (c) Configuration 1 at the Vane 3 mid-span leading edge plane.
Figure 13. Contours of axial turbulence intensity normalized by local velocity, with time-averaged streamlines overlaid for (a) Configuration 2 and (b) Configuration 1. Data is obtained at mid-span plane at the leading edge of Vane 3.

The fluctuating w-velocity component was also obtained in this study. Relative to many turbine inflow turbulence studies that use single component hotwires, this pitchwise fluctuation component is unique, and gives some indication of the anisotropy of the turbulence entering the turbine.

Figure 14 shows w-component turbulence levels, based on mass-averaged turbine inlet velocity, for both clockings. As indicated in the figure, turbulence levels based on fluctuating w-velocity are also larger than 40% upstream of the vane. Comparing between Figures 14 and 12, Configuration 2 shows some similarities in the u- and w-turbulence levels upstream of the vane. However, closer to the vane leading edge, the w-component turbulence level is reduced relative to the u-component level, suggesting that the turbulence becomes more anisotropic near the vane. The Configuration 1 clocking shows less satisfactory agreement between the axial and pitchwise turbulence levels throughout the measurement plane. This is likely due to the anisotropy of turbulence in the near wake of the large ID dilution jet near this location. Table 3 shows the u- and w-component turbulence levels at a point upstream of the vane (x/pitch=0.3, z/pitch=0) for both clocking cases. At this reference location, both cases generate similar levels of turbulence for the two components measured.
Figure 14. Contours of w-component turbulence levels with time-averaged streamlines overlaid for the (a) I=0 case, (b) Configuration 2, and (c) Configuration 1. Data is obtained at mid-span plane at the leading edge of Vane 3.
Table 3. Turbulence levels for both clockings at x/pitch=0.3 and z/pitch=0

<table>
<thead>
<tr>
<th>Clocking</th>
<th>$Tu_m$</th>
<th>$Tw_m$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Configuration 2</td>
<td>0.46</td>
<td>0.44</td>
</tr>
<tr>
<td>Configuration 1</td>
<td>0.46</td>
<td>0.47</td>
</tr>
</tbody>
</table>

Axial ($u$-component) and pitchwise ($w$-component) turbulence levels were extracted as a function of pitchwise direction across the measurement window at x/pitch=0.3. The results are shown in Figure 15 for the two clockings and two turbulence components, as well as for the no-dilution case (I=0). Although there are some differences between the two clockings studied, specifically slightly more variability in turbulence level for Configuration 1 relative to Configuration 2, the overall turbulence levels at this location do not indicate that clocking had a significant impact on turbulence level. The turbulence is generated by the breakdown of the dilution jet coherent structures, which happens relatively independently of the position of the jets relative to the turbine vanes.

Figure 15. Turbulence levels across the measurement plane located at x/pitch=0.3 upstream of the vane at mid-span.

Turbulent integral length scales were calculated from the high speed PIV dataset by performing both temporal and spatial autocorrelations. Only the axial integral turbulent scales are calculated here, so
only the axial component of velocity is utilized in the autocorrelations. For temporally-estimated integral scales, the time record at each PIV interrogation window is used to calculate the temporal autocorrelation $R_{ii}(x, z, \Delta t)$, and Taylor’s frozen turbulence hypothesis is used to determine an integral length scale, $L_{x-t}$. This is performed for each small interrogation region in the PIV measurement plane, producing an integral length scale value for each interrogation region (generally 128 x 128 in the measurement domain).

The procedure to estimate temporal autocorrelations at a point in the flow is illustrated in Figure 16. Figure 16a shows contours of u velocity fluctuations (i.e., instantaneous velocity higher or lower than the time-average value) at an instant in time. A time sequence of data is extracted from the point indicated by the white dot. To reduce noise in the autocorrelation, the entire time sequence (3 seconds, 3000 samples) was broken down into multiple subsets, and the resulting autocorrelation curves from each subset were averaged. The line plot in Figure 16b shows the results of the autocorrelation of these subsets as well as the final average represented by the thick black line. The integral timescale was estimated by integrating the autocorrelation up to the first zero crossing.
Figure 16. Example of (a) a snapshot of axial velocity fluctuation (m/s) and (b) the temporal autocorrelation of axial velocity at the white circle in the contour plot for Configuration 1 and $l=32.7$. Data is obtained at mid-span plane at the leading edge of Vane 3.

Because of the spatially-resolved nature of the flowfield, the spatial autocorrelation of the axial component of velocity was also used to calculate an axial integral length scale. The axial component of velocity was used to determine the spatial autocorrelation $R_{ij}(x,z,\Delta x)$, which was then used to calculate the integral length scale $L_{x-\delta}$. An example of the procedure for this calculation is shown in Figure 17. The spatial autocorrelation was performed for a finite spatial region in the x-direction (see white line on the contour plot). Note that the spatial autocorrelation was not performed over the entire data field, to avoid edge effects as well as any variations very near the vane leading edge. Note that the line plot in Figure 17
does not have a zero crossing, which suggests that the spatial extent of the largest turbulent scales is on the order of the analysis length. This leads to additional uncertainty in the estimation via spatial autocorrelation.

Figure 17. Example of (a) axial velocity fluctuation (m/s) and (b) the spatial autocorrelation of axial velocity (including an exponential fit) along the white line in the contour plot for the Configuration 1 and I=32.7. Data is obtained at mid-span plane at the leading edge of Vane 3.

Figure 18 shows the variation in the axial integral length scale (normalized by OD4 jet diameter) across the pitch of the measurement plane at x/pitch=0.3 upstream of the vane leading edge, calculated using both temporal and spatial methods described earlier. For a given clocking, the two methods of length scale calculation show reasonable agreement with each other, giving some confidence in the ability of this
analysis technique. A quantitative comparison of the average length scale at x/pitch=0.3 is given in Table 4.

Table 4. Average integral axial length scale at x/pitch=0.3 for the ODID4 clocking

<table>
<thead>
<tr>
<th>Method</th>
<th>Lx/OD4 diameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>Temporal autocorrelation (Lx-t)</td>
<td>0.42</td>
</tr>
<tr>
<td>Spatial autocorrelation (Lx-s)</td>
<td>0.38</td>
</tr>
</tbody>
</table>

For both clocking cases in Figure 18, the integral length scale was found to be on the order of the dilution jet diameter, which is similar to previous studies [7]. There is not a clear trend of length scale variation with dilution clocking for this study, although Configuration 1 seems to yield a slightly higher length scale, and Configuration 2 seems to have slightly more variability in integral scale across the pitch. Note, however, that variation in integral scales between cases are within the estimated uncertainties of this quantity.

Figure 18. Comparisons of the axial integral length scale at x/pitch=0.3 and vane mid-span using the temporal and spatial autocorrelations
Chapter 4.

Conclusion

Two-dimensional high-speed PIV measurements at 1 kHz were obtained at the midspan of a turbine vane downstream of a simulated RQL combustor, to study the effects of two dilution hole arrangements. The combustor and vane were representative of commercial aircraft engine geometries, and were scaled up to allow for high measurement resolution. In one dilution hole arrangement (clocking), known as Configuration 1, a large dilution hole was positioned directly upstream of the vane leading edge. In the second arrangement (Configuration 2), the large dilution hole was shifted away from the vane leading edge. A single dilution momentum flux ratio (as well as a no-dilution case) was studied.

Configuration 2 was shown to have a more uniform inlet velocity profile than Configuration 1, although neither were completely uniform in the pitchwise direction as is often assumed during turbine design. This non-uniformity is believed to be the result of aligning the centerline of a singular jet with the vane leading edge. Configuration 2 had the most extreme negative inlet flow angle; Configuration 1 also had a negative inlet angle, but not as severe as Configuration 2. This is believed to be a result of the acceleration of the low-momentum region behind the large OD jet around the vane suction side. Turbulence levels for both clockings were similar, as was expected. This was true also for both recorded components of turbulence, suggesting turbulence isotropy upstream of the vane. The similar levels of turbulence are thought to be a result of the turbulent jets mixing with the crossflow before the vane pressure field, and therefore the clocking effect, acts to distort the flow. This would also explain the similarities in the integral length scale values. Increasing anisotropy between turbulence components near the vane leading edge was influenced by the dilution hole clocking, suggesting that the spatial non-uniformity of the combustor exit mean velocity is important not only for the time-average vane loading, but also for the evolution of the turbulence field in the high-strain region around a vane leading edge. The levels of turbulence (~46%) found
in this combustor configuration are well above previous studies, which would be expected to increase vane leading edge heat transfer and negatively impact the performance of the turbine vane.

This study suggests that the inlet conditions to the turbine for certain combustor types may be more turbulent than previously thought, but also that the high-momentum dilution jets result in a non-uniform velocity profile that can persist into the turbine. Turbulence levels and integral length scales estimated in this experiment could be used in the correlations mentioned above to predict heat transfer augmentation on a 1st vane. These correlations, however, do not take into account non-uniform velocity distributions upstream of the vane. Clearly it is important to consider the spatially- and temporally-resolved influence of the combustor exit flow on the 1st vane in the quest to improve gas turbine efficiency and reliability. Future studies should investigate the relationship between the dilution hole placement in both the pitchwise and streamwise directions relative to the turbine vanes, and provide more details of the spatial distribution of velocity, turbulence and integral length scales entering the turbine.
References


