Combustor Dilution Hole Placement and Its Effect on the Turbine Inlet Flowfield

Michael Leonetti, Stephen Lynch, and Jacqueline O’Connor
Pennsylvania State University, University Park, Pennsylvania 16802
and
Sean Bradshaw
United Technologies—Pratt & Whitney, East Hartford, Connecticut 06108
DOI: 10.2514/1.B36308

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. In this study, a dilution hole configuration was scaled from a rich-burn–quench–lean-burn 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 and turbulence level were dependent on the position of the dilution jets relative to the vane.

Nomenclature

\begin{align*}
C_{Ax} &= \text{vane axial chord} \\
C_p &= \frac{(P_i - P_{in})}{0.5\rho U^2}, \text{nondimensional static pressure} \\
D &= \text{dilution jet diameter} \\
I &= \frac{(pU^2)}{\rho}, \text{momentum flux ratio} \\
L_{c-x} &= \int \frac{R_z(x, z, \Delta x)}{\Delta x} \text{d}x, \text{spatial axial integral length scale} \\
L_{c-t} &= \int \frac{R_z(x, z, \Delta t)}{\Delta t} \text{d}t, \text{temporal axial integral length scale} \\
\bar{m} &= \rho U, \text{mass flow rate} \\
P &= \text{pressure} \\
R_0(x, z, \Delta t) &= \left(\frac{u(x, z, t) \cdot u(x, z, t + \Delta t)}{\bar{U}_m^2(x, z)}\right), \text{temporal autocorrelation coefficient} \\
R_0(x, z, \Delta t) &= \left(\frac{u(x, z, t) \cdot u(x + \Delta x, z, t)}{\bar{U}_m(x, z)}\right), \text{spatial autocorrelation coefficient} \\
S &= \text{test-section span} \\
s &= \text{surface distance along vane} \\
T_u_{loc} &= \frac{(u_{rms}/U_{loc}) \cdot 100}, \text{axial component of turbulence intensity normalized by the local velocity magnitude, \%} \\
T_u_{m} &= \frac{(u_{rms}/U_{m-avg}) \cdot 100}, \text{axial component of turbulence levels normalized by the turbine inlet mass-average velocity magnitude, \%} \\
T w_m &= \frac{(u_{rms}/U_{m-avg}) \cdot 100}, \text{pitchwise component of turbulence levels normalized by the turbine inlet mass-average velocity magnitude, \%} \\
U &= \text{velocity magnitude} \\
U_{jet-plenum} &= \sqrt{\frac{2}{\rho}} \frac{P_{plenum} - P_{in}}{\rho}, \text{dilution jet velocity determined using plenum and mainstream pressure data} \\
U_{jet-pilot} &= \sqrt{\frac{2}{\rho}} \frac{\Delta P}{p_{jet}}, \text{dilution jet velocity determined using pilot probe data} \\
U_m &= \text{velocity magnitude normalized by the turbine inlet mass average velocity} \\
U_{m-avg} &= \frac{\sum n_{dilution jet}}{\rho \cdot W}, \text{mass-averaged turbine inlet velocity magnitude} \\
u &= \text{axial component of velocity} \\
v &= \text{spanwise component of velocity} \\
w &= \text{pitchwise component of velocity} \\
x &= \text{axial direction} \\
y &= \text{spanwise direction} \\
z &= \text{pitchwise direction} \\
\alpha &= \text{incidence angle} \\
\nu &= \text{kinematic viscosity} \\
\rho &= \text{density} \\
\langle \rangle &= \text{time average of a quantity} \\
\text{jet} &= \text{jet property} \\
\text{loc} &= \text{normalized by a local velocity magnitude} \\
m &= \text{normalized by a turbine inlet mass-average velocity magnitude} \\
m-avg &= \text{mass-averaged value} \\
rms &= \text{root mean square} \\
\infty &= \text{mainstream property}
\end{align*}

I. 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 nonuniform flowfield (if mixing is insufficient) that enters the turbine section. This oncoming flowfield can be very detrimental to the turbine vane durability because the gas temperatures can exceed the vane melting temperature. A current trend in commercial aviation gas turbines is increasingly smaller engine cores to achieve ultrahigh bypass ratios for high propulsive efficiency. Thus, combustors continue to shorten in length, potentially positioning dilution jets closer to the downstream vanes. Although 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 flowfield 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.

II. Relevant Literature

Many experimental and numerical studies have investigated combustor and turbine flowfields separately, but few have both spatially and temporally 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 and 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 a 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 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 and 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 turbulence levels at the combustor-turbine interface of approximately 35% and length scales of up to 25% of the vane chord length. Vakil and Thole [6] used a nonreacting cold-flow combustor with both dilution and film cooling holes that had measured turbulence levels of 20%. 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 nonuniform thermal fields. Barringer et al. [7] conducted a similar experiment, which yielded slightly lower turbulence 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 and disperse film cooling, particularly at the stagnation point on a vane (Ames [9], Gandavarapu and Ames [10], Ames [11], Nasir et al. [12], Radomsky and Thole [13]).

Most prior studies investigating the effects of high freestream turbulence on heat transfer augmentation have used simulated turbulence from bar grids. Van Fossen et al. [14] 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. In their work, a correlation was proposed to predict the effect of augmentation of stagnation heat transfer; however, this correlation was based on isotropic turbulence, which may not be appropriate for the vane in an engine. In fact, their correlation underestimated the stagnation heat transfer by up to 11% for turbulence from a parallel-wire grid, which generated significant turbulence anisotropy. Ames [2] considered higher levels of turbulence and their effect 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 $l_u$, which incorporates turbulent kinetic energy and length scale, had a significant impact on the stagnation and pressure surface heat transfer. Gandavarapu and Ames [10] found that grid-generated turbulence correlations underpredicted vane heat transfer augmentation, whereas their scaling parameter, based on turbulence level, Reynolds number, and turbulent length scale, overpredicted heat transfer augmentation for a very-large-diameter leading edge (i.e., high Reynolds number). In general, correlations can be found to bound vane stagnation heat transfer augmentation at high turbulence levels but may not be appropriate for other situations, 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 because modeling both can be computationally intensive. Experimental results have shown that the flowfield at the combustor outlet is highly turbulent and spatially nonuniform, 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. [15] showed through experimental and computational studies that the nozzle guide vane’s (NGV) potential field has an effect on the upstream combustor flow. Cha et al. found that the NGV’s impact occurs well before the combustor–turbine interaction plane where many combustor-only studies end. Salvadori et al. [16] performed two Reynolds-averaged Navier–Stokes (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 decoupled simulation did a poor job of predicting the flow entering the vane section. This resulted in a turbine simulation that overpredicted 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 and Thole [17] showed that the use of a two-dimensional turbine inlet boundary condition near the end wall, as is typically done in practice, was inaccurate; instead, the exit flowfield from a realistic combustor exhibited three-dimensional behaviors with nonuniformities in temperature, pressure, and velocity.

Insinna et al. [18] also used a coupled combustor/turbine simulation, which found similar nonuniformities 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. [19] compared simulations and experiments from an annular combustor--turbine rig, which included dilution jets and reacting flow. Each steady RANS turbulence model predicted different temperature profiles at the turbine inlet plane and underpredicted jet mixing as well as lateral spreading. Cha et al. [5] noted that higher-fidelity modeling (large-eddy simulation) more accurately predicted the turbulence intensities found at the combustor--turbine interface in their experiment. Simply modeling the combustor and turbine together may not be enough to ensure accurate results; higher-fidelity unsteady models are needed to accurately predict the large turbulent structures stemming from the unsteady combustor flowfield.

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.

III. Experimental Setup

This experiment used a simulated combustor and scaled vanes in a low-speed large-scale wind tunnel shown in Fig. 1. This wind tunnel has been used in previous investigations of combustor and other dilution flow studies (Vakil and Thole [6], Barringer et al. [7], Stitzel and Thole [17]). For those studies, the combustor geometry was based on older technology and had a significant flow area convergence upstream of the vane, which was eliminated for this study. Also, the previous dilution studies were at a larger scale and thus modeled a single combustor sector with two rows of dilution, whereas this study models two sectors with a single row of dilution.

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, whereas the two bypass flow sections were used 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. [20]. The vane test section has five vanes, an inlet span height of 1.912C_{ax}, a pitch of 1.215C_{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 (Perdochizzi [21], Hermanson and Thole [22]). 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 et al. [23] and Arts and De Rouvroit [24]. Note that this study is focused on the leading-edge region of the first vane, where velocities are low even for transonic vanes (Nix et al. [25], Barringer et al. [26]). Finally, the experiment was conducted without the addition of fuel and reactive products, although studies such as those conducted by Zimmerman [27] and Moss and Oldfield [28] have shown that turbulence levels values downstream of combustion were unaffected by the combustion process.

For this study, a commercially relevant RQL-style combustor was geometrically scaled for the large wind tunnel and inserted upstream of the vane test section. Because of vane geometry constraints, the Holdeman parameter (Holdeman [29]) of the scaled combustor geometry was smaller than the optimum of 5.0 (Holdeman [29]) for combustors with offset dilution hole centers, which has resulted in some underpenetration of the dilution jets relative to an optimum configuration. The primary feature scaled from the engine condition was the dilution hole geometry; other features of the combustor such as liner cooling were not included in this work. 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. The study by Cha et al. [15] for a similar RQL geometry also indicated little effect of upstream swirl on dilution jet trajectory.

The momentum flux ratio of the dilution was matched to a representative engine condition. Momentum flux ratio was determined to be the most significant aerodynamic parameter because this will determine the jet’s trajectory, as discussed earlier. Because 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; however, the jet penetration is the primary factor in the jet mixing behavior. Note also that, in the wind tunnel, the vane cascade geometry is planar (not annular), and so the inner diameter (ID) and outer diameter (OD) walls in the wind tunnel have the same arc length. In the wind-tunnel implementation, the bottom wall of the tunnel was designated as the OD end wall, and the top wall was designated as the ID end wall. 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 Fig. 2. The dashed line in Fig. 2b shows the measurement plane that was investigated in this study.

The simulated combustor consisted of two full combustor sectors, where each sector had four 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 eight dilution holes each across the entire pitch of the tunnel. The dilution hole centerlines were located 1.77C_{ax} upstream of the vane. The combustor was not designed with flow convergence because it was determined during the scaling analysis that the flow convergence through the remainder of the combustor up to the vane was negligible; however, Fig. 2 indicates that the vane does have upper- and lower-wall convergence to replicate engine conditions.

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

![Fig. 1 Large-scale low-speed wind-tunnel facility.](image-url)
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.

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 jet was expected to impact the opposing end walls. 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 Fig. 1).

The flowfield measurements were taken with a high-speed particle image velocimetry (PIV) system. The flow was seeded with 1 μm particles of diethyl 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 an 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 used a 60 mm lens and had a 1024 × 1024 pixel resolution and a capture frequency of 2000 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 $\Delta t = 30 \, \mu s$ 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 multipass method was used during PIV calculation with decreasing window size. The first two passes were done with a 64 × 64 window size and 50% overlap. The window size then decreased to 16 × 16 at 50% overlap through four passes. Minimal postprocessing 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 resulting from the full processing routine were the first choice from the respective cross-correlation. 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 statistical analysis 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 of vane 3, at its midspan. The measurement plane is the turbine inlet radial plane that captures two-dimensional velocity ($u$ and $w$) ahead of the vane 3 leading edge, as shown in Figs. 2 and 3. The measurement plane is 0.61 $C_{ax}$ by 0.61 $C_{ax}$ in dimension. Two-dimensional PIV was chosen over stereoscopic PIV due to the limited optical accessibility in front of the vanes.

A. 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 six subsets to be used in a precision uncertainty analysis. Precision uncertainty was calculated using the method described by Moffat [29] 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, due to the sensitivity of integral scale determination on the sample size as well as the significant temporal variation of dilution jet wake. Percent precision uncertainty is reported at a point aligned with the leading edge of the vane (z/pitch = 0), at 0.3 pitch upstream ($x$/pitch = 0.3), and results are shown in Table 1.

The total uncertainty, which considers both bias and precision uncertainty, 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 conservative estimate for the bias uncertainty calculation (Wieneke [30]). Note that the bias uncertainty is a significant portion of the total uncertainty in the measurements.

B. 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. Experimental results were compared to results obtained from a periodic computational fluid dynamics (CFD) simulation (Gibson et al. [20]) to ensure that the flow entering all vane passages were correct before introducing the dilution flow. The vane pressure loading can be seen in Fig. 4.

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:

$$U_{jet−plenum} = \sqrt{\frac{2 \times (P_{\infty} − P_{plenum})}{\rho_{\infty}}}$$

(1)

Table 1 Precision uncertainty (percent of mean), at $x$/pitch = 0.3, $z$/pitch = 0

<table>
<thead>
<tr>
<th>Case</th>
<th>$U$</th>
<th>$T_{u\infty}$</th>
<th>$T_{w\infty}$</th>
<th>$L_{u\infty}$</th>
<th>$L_{w\infty}$</th>
<th>$I$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$I = 0$</td>
<td>0.3</td>
<td>3.4</td>
<td>1.6</td>
<td>11.8</td>
<td>29.1</td>
<td></td>
</tr>
<tr>
<td>Configuration 1</td>
<td>3.3</td>
<td>4.7</td>
<td>3.6</td>
<td>7.4</td>
<td>13.4</td>
<td></td>
</tr>
<tr>
<td>Configuration 2</td>
<td>2.8</td>
<td>1.5</td>
<td>1.1</td>
<td>5.3</td>
<td>17.3</td>
<td></td>
</tr>
</tbody>
</table>

Table 2 Total uncertainty (percent of mean) of velocity magnitude, at $x$/pitch = 0.3, $z$/pitch = 0

<table>
<thead>
<tr>
<th>Case</th>
<th>Uncertainty, %</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>

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

$$I = \frac{\rho U_j^2}{\rho \bar{U}_j^2}$$

(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 et al. [20], with the time-average result from the PIV data taken in the measurement plane shown in Fig. 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 centerline velocities). The mass flow contribution of each hole is then included in the total mass flow downstream of dilution injection, to determine the average turbine inlet velocity:

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

(3)

Good agreement is found in Fig. 5 between the CFD and PIV measurement for both magnitude of velocity as well as 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 region of invalid data at the bottom of Fig. 5b is larger than the actual vane leading edge.

Another check performed was the repeatability and statistical convergence of the measurements. At least three data sets were obtained for each flow condition and dilution clocking. Because of camera memory limitations, the maximum amount of continuous samples was limited to 1000 in each data set. Figure 6 shows a comparison of the time average of each of the three data sets, which show good agreement.
Although the preceding comparison 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 Fig. 7. The average of three sets was deemed sufficient for stationarity in the fluctuating velocity component (presented as turbulence level in the figures).

IV. Results

A. Time-Averaged Flow Structure

To help orient the reader on the flowfield generated by the dilution jets upstream of the vane leading edge, Fig. 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 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.

Experimental measurements of the time-averaged normalized velocity magnitude are shown in Fig. 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 Fig. 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.

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 $\alpha$ of 0 deg until the vane pressure field begins to turn the flow around the vane. The remaining
two contour plots in Fig. 9 show the results for the two clockings investigated in this study, at a momentum flux of $I/0.136$. As described earlier, the core of the dilution jets is expected to penetrate past this plane upstream of this measurement window.

For configuration 2 (Fig. 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 Fig. 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, Fig. 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.

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 Fig. 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 ($x/C_{ax} = 0.365$). 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 $-7.9\,\text{deg}$ with an average across the pitch of the measurement plane of $-5.2\,\text{deg}$. 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 $-19.4\,\text{deg}$ with an average of $-15.2\,\text{deg}$ across the measurement plane. This negative inlet angle likely has a significant impact on the location of the vane stagnation and might result in a small suction-side separation on this airfoil, although the density of static pressure taps on the airfoil was not sufficient to determine this.

Fig. 7 Axial turbulence level from a) one data set, b) two combined sets, and c) three combined sets.

Fig. 8 Computational prediction of $U_m$ through the centerline of the dilution jets in configuration 1 (unpublished).

Temporal variations of the turbine inlet flow angle were also investigated because the dilution flow is naturally unsteady. A common design practice in industry is to specify circumferentially averaged velocity and temperature profiles from the combustor, as
inlet conditions for the turbine. To indicate the significant increase in overall flowfield unsteadiness with dilution, we spatially averaged the local flow angle in the pitch direction (circumferential direction in an engine) at each time instance, to provide a spatially averaged but instantaneous inlet flow angle. The spatial averaging process will result in less extreme temporal flow angle variation than at a single location in the flowfield but will give a picture of incoming global flow variation. 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 $+/- 40$ deg. 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 suction side.

B. 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 level $T_u$ values are shown in Fig. 12, where only the rms of the $x$ component of the velocity was used. Axial velocity rms was normalized by the mass-averaged turbine inlet velocity in this figure 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 7% at the turbine leading edge based on grid turbulence correlations (Roach [31]). 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 for typical aeroengine combustor configurations. This is likely due to the low Holdeman parameter of the current combustor geometry as described earlier, which is manifested as underpenetration of the dilution jets and increased unmixedness (Holdeman et al. [3]). However, note that the reported value of turbulence at the combustor exit 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

---

**Fig. 9** Normalized time-averaged velocity magnitude with streamlines for a) $I = 0$, b) configuration 2, and c) configuration 1.

**Fig. 10** Time-averaged inlet flow angle across the measurement window at $x/pitch = 0.3$ upstream of the vane.
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 nonuniform distribution of velocity magnitude in Fig. 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 that 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 Fig. 2) 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 Fig. 9.

The fluctuating \(w\)-velocity component was also obtained in this study. Relative to many turbine inflow turbulence studies that use single-component hot wires, 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 Figs. 12 and 14, 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.

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 Fig. 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.

Turbulent integral length scales were calculated from the high-speed PIV data set by performing both temporal and spatial autocorrelations.
Only the axial integral turbulent scales are calculated here, and so only the axial component of velocity ($x$-direction velocity) is used 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 $128 \times 128$ regions in the measurement domain).

The procedure to estimate temporal autocorrelations at a point in the flow is illustrated in Fig. 16, for configuration 1. Figure 16a shows contours of instantaneous axial velocity fluctuations. A time sequence of data is extracted from the $x, z$ point indicated by the white dot, and an autocorrelation is performed on the time sequence. To reduce noise in the autocorrelation, the entire time sequence (3 s, 3000 samples) was broken into multiple subsets, and the resulting autocorrelation curves from each subset were averaged. The line plot in Fig. 16b shows the various autocorrelation coefficients of

![Image](https://example.com/fig16.png)

**Fig. 12** Axial turbulence level, with time-averaged streamlines for a) $I = 0$, b) configuration 2, and c) configuration 1.

![Image](https://example.com/fig13.png)

**Fig. 13** Axial turbulence level normalized by local velocity for a) configuration 2, and b) configuration 1.
these subsets as a function of temporal lag, as well as the final average correlation coefficient, represented by the thick black line. The integral time scale was estimated by integrating the autocorrelation up to the first zero crossing and converted to a length scale by multiplying by the time-average $x$ velocity at that location.

Because of the spatially resolved nature of the flowfield, the spatial autocorrelation of the axial component of velocity could also be used to calculate an integral length scale. The axial component of velocity in the axial direction was used to determine the spatial autocorrelation $R_{ii}(x, z, \Delta x)$, per the procedure described by McManus and Sutton [32]. An example of the procedure for this calculation for configuration 1 is shown in Fig. 17. The spatial autocorrelation was performed for only a finite spatial extent in the $x$ direction (see the extent of the white line on the contour plot, from $x$/pitch = 0.45 to 0.05), to avoid invalid data at the very edge of the measurement domain as well as any variations very near the vane leading edge. Radomsky and Thole [33] indicate that the strain field at a vane leading edge results in a local decrease in integral length scale very close to the stagnation. The axial velocity fluctuation at the $(x, z)$ point shown in Fig. 17a was multiplied by the fluctuation at an axial displacement of $x + \Delta x$ for each time instance, and the result was time-averaged and divided by the product of the axial velocity rms at $(x, z)$, and $(x + \Delta x, z)$ to obtain the correlation coefficient $R_{ii}$ (see the equation in the Nomenclature). Figure 17b shows an example spatial autocorrelation corresponding to the $x, z$ location in Fig. 17a, where $\Delta x$ is the spatial displacement relative to $x, z$. If the correlation coefficient reached a zero crossing, it was integrated to that point to calculate the integral length scale $L_{x=z}$. Note that, in Fig. 17a, the autocorrelation does not quite reach a true 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. In those cases, a least-squares exponential fit was performed on the autocorrelation per Eq. (4) [32]:

$$R_{ii, fit} = a \cdot e^{(-b+\Delta x)}$$

Table 3 Turbulence levels at $x$/pitch = 0.3, $z$/pitch = 0

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

Fig. 14 $w$-component turbulence level with time-averaged streamlines for a) $I = 0$, b) configuration 2, and c) configuration 1.

Fig. 15 Midspan turbulence levels across the measurement plane at $x$/pitch = 0.3 upstream of the vane.
where $a$ and $b$ are fit coefficients. Then, the integral scale $L_{x,t}$ is the inverse of the $b$ coefficient. The length scales estimated from an exponential fit and from integration of the autocorrelation curve generally agreed well with each other.

Figure 18 shows the variation in the axial integral length scale, normalized by large jet diameter, across the pitch of the measurement plane for the two analysis methods described (temporal autocorrelation and spatial autocorrelation). Note that the temporal autocorrelation can provide estimates of $L_{x,t}$ throughout the entire measurement domain because each PIV interrogation window provides a time signal, but the spatial autocorrelation is performed over a finite spatial window in the $x$ direction and thus can only provide distinct values for $L_{x,s}$ in the pitchwise direction. In Fig. 18, $L_{x,t}$ is evaluated at $x/pitch = 0.45$ for comparison to $L_{x,s}$. For a given clocking, the two methods of integral length scale calculation show somewhat reasonable agreement with each other. A quantitative comparison of the pitchwise-average length scale at $x/pitch = 0.45$ for configurations 1 and 2 is given in Table 4.

For both cases in Fig. 18, the integral length scale was found to be on the order of the dilution jet diameter, which is similar to previous studies (Barringer et al. [7]). There is not a clear trend of length scale variation with dilution clocking for this study, although the two configurations seem to be out of phase, which might be expected from the different dilution hole positions. Note that variation in integral scales between cases is within the estimated uncertainties of this quantity.

<table>
<thead>
<tr>
<th>Table 4</th>
<th>Average axial integral length scale at $x/pitch = 0.45$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Method</td>
<td>Configuration 2</td>
</tr>
<tr>
<td>Temporal autocorrelation: $L_{x,t}/D$</td>
<td>0.40</td>
</tr>
<tr>
<td>Spatial autocorrelation: $L_{x,s}/D$</td>
<td>0.38</td>
</tr>
</tbody>
</table>
V. Conclusions

Two-dimensional high-speed particle image velocimetry measurements at 1 kHz were obtained at the midspan of a turbine vane downstream of a simulated rich-burn–quench–lean-burn 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 was completely uniform in the pitchwise direction, as is often assumed during turbine design. This nonuniformity 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 outer diameter 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 nonuniformity 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 nonuniform velocity profile that can persist into the turbine. Turbulence levels and integral length scales estimated in this experiment could be used in the correlations available in literature to predict heat transfer augmentation on a first 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.

Acknowledgment

The authors would like to thank United Technologies — Pratt & Whitney for their financial and technical support of this work.

References


K. Frendi
Associate Editor