Experimentally Determined External Heat Transfer Coefficient of a New Turbine Airfoil Design at Varying Incidence Angles

Kyle F. Chavez¹*, Gavin R. Packard¹, Tom N. Slavens², David G. Bogard¹

Abstract
Predicting and measuring heat transfer coefficients of hot gas path turbine components are extremely important tools for gas turbine designers. Accurate prediction of heat transfer coefficients with CFD is strongly dependent upon calibration and validation using experimental measurements. Since turbine components often handle a range of inlet flow incidence angles during dynamic flight situations, understanding the sensitivity of heat transfer coefficients to a range of inlet conditions is important. In the past, researchers have performed CFD predictions for turbine airfoils and vanes at off-nominal incidence angles, but few researchers have directly measured the heat transfer coefficients, and then only at shallow, off-nominal angles. This research program filled a void by evaluating incompressible CFD predictions with direct measurements of heat transfer coefficients for appreciably off-nominal incidence angles at both high and low turbulence. Particle Image Velocimetry was used to verify the incident flow angle. The high turbulence condition was generated using a turbulence grid to produce \( Tu = 5\% \) at the leading edge of the test article, whereas the low turbulence condition had \( Tu = 0.6\% \). The test facility was adjusted so that the pressure distribution around the model airfoil matched the pressure distribution predicted by an aerodynamic CFD model. IR thermography and an airfoil model equipped with a constant heat flux surface were used to measure the heat transfer coefficient. This process was repeated for each incidence angle measured. The results quantified the effects of inlet angle upon the heat transfer coefficient and the accuracy of the SST-\( k\omega \), SST-Transition, and \( k-kl-\omega \) models used to predict the experimental results.

Keywords
Heat Transfer Coefficient — Incidence Angle — Experimental

¹ Department of Mechanical Engineering, University of Texas at Austin, Austin, United States
² Pratt & Whitney, Hartford, United States
*Corresponding author: kyle.f.chavez@utexas.edu

INTRODUCTION
The ever-growing demand for increased efficiency of hot gas-path components for gas turbines has become a driver for the development of advanced aerodynamic profiles, film-cooling schemes, and material durability. The research in these areas has allowed turbine designers to significantly increase the capabilities of modern day turbines. However, these advanced turbine designs are relying more and more upon commercially available computational fluid dynamics (CFD) software or ‘in-house’ CFD to predict key aerodynamic and heat transfer parameters. Although the external heat transfer coefficient (HTC) distribution is a critical parameter for designers, the HTC distribution is still more difficult to predict with CFD than aerodynamics. Therefore, it is important to measure the HTC distribution directly in order to further improve the predictive capabilities of CFD. This is especially true when the scenario is more complex than usual, such as when unconventional designs, off-design conditions, and varying turbulence levels are involved.

Beyond the many studies measuring only HTC distributions, some researchers have sought to measure the effects of the external flow field on the HTC distribution while keeping the inlet incidence constant. For example, Arts et al. [1] presented results of varying Mach number, Reynolds number, and turbulence intensity on the HTC distribution, measured with thin film gauges. Carullo et al. [2] studied the effects of turbulence length scale in addition to turbulence intensity and Reynolds number, also with thin film gauges. These studies showed an increase in heat transfer coefficients with increasing Reynolds number, increasing freestream turbulence level, and decreasing length scale. Both studies also indicated an earlier transition with increasing Reynolds number.

A small number of studies have also investigated the effects of incidence angle on the HTC distribution. The majority of these measurements are made with a low spatial
resolutions due to the employed measurement techniques. In addition to the effects listed previously, Arts et al. [1] studied the effects of incidence angle changes from -14° to +11° on a SNECMA RS1S airfoil profile using midspan gauges. They found that the incidence angle had a noticeable effect on the transition point on the suction side with earlier transition occurring for positive incidence angles and later transition occurring negative incidence angles. There was little effect on the pressure side except for the -14° case which caused a spike in the HTC immediately downstream of the stagnation point, which the authors speculated was due to a small separation bubble.

Camci and Arts [3] studied incidence angle affects at -10° to +15° off design on a film-cooled airfoil model with mid-span thin film temperature gauges using a transient analysis to deduce heat transfer rates. They also found that the incidence angle altered the transition location on the suction side, with increasing positive incidence angles causing earlier transition. On the pressure side increasing positive incidence angles caused an extended laminar and transitional region resulting in decreased the HTC.

More recently Giel et al. [4] provided high-detail heat transfer coefficient measurements made with a constant heat flux surface and liquid crystal surface temperature measurements. Measurements were made for incidence angles of 0° and ±5° off-design conditions. However interpretation of the incidence angle affects was complicated by a simultaneous change in inlet Reynolds number, with many of the effects being attributed to the change in Reynolds number. There is no literature which presents measurements of the heat transfer coefficient distribution in high-resolution on a linear cascade at incidence angles larger than ±5°.

Dunn [5] reviewed the current state of the HTC predictive capability of CFD in 2001. Dunn found that many improvements are still required to bring the HTC predictive capability up to that of the aerodynamic predictive capability. Since then, a number of measurement and CFD comparisons have been made, oftentimes with improved or higher order turbulence models. In addition to the high-resolution HTC measurements made, Giel et al. [4] predicted the HTC distribution with a multistage Runge-Kutta 3D Navier Stokes code described by Chima and Yokota [6] and found that, while the trends of the experimental and simulated data matched, the predictive capability of the code was wholly unsatisfactory for the parameters studied. Dees et al. [7] compared HTC measurements to an SST-Transition model prediction generated with a Fluent RANS solver, and was able to predict small portions of the showerhead and pressure-side HTC within experimental uncertainty, as well as the overall trends of the HTC distribution. Dees et al. also utilized the Wilcox k-ω model, but with less success. However, Tallman et al. [8] showed that using a modified 3D, compressible, RANS solver with a k-ω model, GE's TACOMA platform could be used to predict HTC distributions quite well for high-pressure vanes and airfoils, and fairly well for low-pressure vanes. Furthermore, Luo et al. [9] performed a comparative study of various turbulence models on a turbine airfoil and end wall, and found that 3D CFD predictions with the SST model has better accuracy than the k-ε, realizable k-ε, and V2F models, but also agreed that additional studies are needed to address more complex scenarios than those presented in the study.

Prior to the current study, there have been few publications of the sensitivity of the external HTC distribution of airfoils to a combination of turbulence and incidence angle variations. The work presented here compares the experimental effects of turbulence, from 0.6% to 5.0%, as well as a nominally +9° change in incidence angle on a scaled up model of a new turbine airfoil design to incompressible pressure-based CFD solutions. The model used is a scaled up mid-span extrusion of the new airfoil design modified for low speed testing to allow for pressure distribution matching within a linear cascade. In order to match the flow field to turbine operating conditions, the true inlet incidence angle was measured with PIV, and the inlet Reynolds number, turbulence level, turbulence length scale, and stagnation line were all matched within a pre-determined accuracy.

The results from this study quantified the effects of inlet angle and turbulence intensity on the heat transfer coefficient and the accuracy of the SST-κε, SST-Transition, and k-kl-ω models used to predict the experimental results and also verified the functionality of the new wind tunnel designed and manufactured for this study. The study did not investigate additional flow phenomena which would further complicate the flow conditions, such as wake passing or film fouling.

1. EXPERIMENTAL SETUP

Pressure distribution, heat transfer coefficient, and velocity field measurements were performed on the new airfoil model in a closed-loop wind tunnel at the Turbulence and Turbine Cooling Research Laboratory (TTCRL) located at the University of Texas at Austin. The wind tunnel test section contains two turning vane stages upstream of the test models which redirect the upstream flow allowing for measurements to be performed at varying incidence angles shown in Figure 1. A turbulence grid can be installed between the second stage of turning vanes and the test cascade to generate the required turbulence levels at the leading edge of the test airfoil.

The wind tunnel test section measures 127 cm by 54.9 cm, and is a 2.33 passage linear cascade with 2 full airfoil models and a 1/3 airfoil model. The center airfoil model is referred to as a test article as it is the airfoil model used to make experimental measurements. The 1/3 airfoil model is attached to a flexible wall, which can be adjusted to force precise pressure distributions on the center test airfoil. On the inner side of the test section, an adjustable blockage is present which acts as a loss mechanism, affecting the mass flow rate through the inner passage. An inline blower is attached to the outer passage to affect the mass flow rate through the outer passage while allowing for direct optical access to the suction side of the airfoil model. These controls on the mass flow
rates around the dummy airfoils are used to set the stagnation lines on the cascade.

Upstream of the test section, two linear turning vane cascades were present to redirect the flow. The first stage of turning vanes directed the flow out of the contraction nozzle and into the second stage, while the second stage directed the flow into the test section. The rig was designed and built to handle a range of incidence angles that might occur for a rotating turbine airfoil. For the current study the airfoil was designed to operate with a nominal incidence angle of -30° relative to the axial direction (0°). For the high turbulence case, tests were conducted using nominally -30° and -21° incidence angles, which correspond to 0° and +9° relative to the design engine incidence angle. For the low turbulence case the incidence angles were nominally 34° and 25°, or -4° and +5° relative to the design angle. The incidence angles for low and high turbulence cases were not the same due to additional turning caused by the turbulence grid. Several extendable walls were present downstream of the second turning vane stage, but upstream of the linear cascade, which compensated for the additional wall length needed for incidence angle adjustments in the tunnel. The turning vanes and adjustable walls were both present to allow the whole test section, including the airfoil models, blockages, blower, cooling loop (not used in these tests), and all accompanying instrumentation to remain stationary over the designed range of incidence angle changes.

Two airfoil models were manufactured in order to measure both the pressure distribution and heat transfer coefficient. These two airfoil models were known as Test Article A (the pressure distribution model) and Test Article B (the heat transfer coefficient model). These models had top and bottom lids for a modular cartridge-type removal and installation system, with which different airfoil and vane models can be equipped to facilitate a quick and easy reconfiguration of the linear cascade.

1.1 Pressure Measurement Airfoil
Test Article A was a scaled up version of the new airfoil design and was made of a closed-cell polyurethane foam with axial chord length of $C = 0.355$ m. The test article contained a total of 34 pressure taps, with 22 being located at 50% span, 6 located at 40% span, and 6 located at 60% span. The taps consisted of 0.83 mm inner diameter steel hypodermic tubing routed in to the core of the model, attached to EVA tubing, and then fed out the top of the model to pressure transducers.

1.2 Heat Transfer Coefficient Measurement Airfoil
Test Article B was also manufactured from a closed-cell polyurethane foam (0.044 W/(m·K)), which was then covered with a 0.10 mm thick, 30.5 cm wide, and 113 cm long 301 stainless steel shim. The ends of the shim were fastened between 1.6 mm X 12.7 mm copper bus bars to provide a uniform heat flux along the surface of the airfoil model. The bus bars were embedded into the polyurethane model at a location as far downstream as possible so as to minimize any flow field disruptions.

2. Test Methodology
The following pre-test steps were performed for each incidence angle:

• Using CFD calculate the airfoil pressure distribution and stagnation line that would occur for an infinite cascade of airfoils with the incidence angle expected for this test.
• Adjust the downstream walls, blockage, and inline blower to set the desired stagnation line position and pressure distribution.
• Measure flow field uniformity, inlet flow incidence angle, and turbulence level with PIV and hotwire anemometry.
• Measure the change in incidence angle with the turbulence grid installed using PIV.
• Verify the turbulence level created at the leading edge by the grid with hotwire anemometry.

Once the pre-test steps were performed, actual test data was generated by performing the following steps:

• Use CFD to recalculate the airfoil pressure distribution for an infinite cascade using the measured incidence angle.
• Make a second adjustment to the downstream walls, blockage, and inline blower to match of the corrected pressure distribution and stagnation line positions.
• Install Test Article B with 12 external surface thermocouples attached.
• Calibrate IR cameras by recording images at the thermocouple locations with four IR cameras while varying surface temperatures by adjusting the current delivered to the heat flux foil.
• Remove the thermocouples from the external
surface of Test Article B and measure 2D surface temperature distributions at steady state.

- Convert surface temperature measurements to heat transfer coefficients with equations 3 through 6.

2.1 Turbulence Grid

The turbulence grid consisted of an array of vertical cylindrical rods which spanned the test section upstream of the test cascade with a periodic spacing of 19.0 mm. The diameter of the rods was \( b = 4.76 \) mm resulting in a grid solidity of 0.25. The grid design was based on prior work done in our laboratory by Mosberg [10] who showed that turbulence generated by this type of grid decayed according to the following correlation by Roach [10]:

\[
Tu = A \left( \frac{x_0}{b} \right) \tag{1}
\]

where \( A \) is a correlation constant, and \( x_0 \) is the distance from the grid in the flow direction. Preliminary measurements for this test program established the value for the correlation coefficient for this grid as \( A = 0.75 \). For the high turbulence cases in this test program, the turbulence grid was positioned to obtain \( Tu = 5.0 \% \) at the leading edge of the test article.

In addition to turbulence intensity, the turbulence grids were designed to obtain an integral length scale of \( \lambda_T/C = 0.05 \). For low turbulence tests, the turbulence grid was not installed, and the turbulence levels were measured to be \( Tu = 0.6\% \).

It is also important to note that during this phase of testing, measurements were made to assess the uniformity of the flow leading in to the test section. This included both turbulence levels and velocity distributions. Very small, periodic, pitchwise non-uniformities due to the turning vanes were present, but the increase in \( Tu \) and the decrease in velocity due to the turning vane wakes were both <5% of their measured values a short distance downstream of the turning vanes. Both the turbulence and velocity non-uniformities were very small relative to the size of the model being measured, and both were suppressed significantly due to the expected velocity nonuniformity experienced in the test section as the flow approached the cascade of airfoils (as verified with PIV). In addition to these effects, the presence of the turbulence rods for the high turbulence case wiped out the non-uniformities, as the turbulence rods generated turbulence significantly stronger than that generated from the turning vanes. Finally, a separate experiment was performed with the wind tunnel in its neutral position. In this experiment, HTC data was collected both without turning vanes and with rods installed which generated identical wake profiles and turbulence levels as the turning vanes did when the tunnel was in a non-neutral position. In the case of these tests, the heat transfer coefficient magnitudes remained well within measurement uncertainty, and there were no measureable effects due to the turning vane wakes.

2.2 Incidence Angle

As can be seen in Figure 1, the mainstream passed through the turbulence grid at an angle different than the normal to the plane of the grid. This caused a turning of the mainstream towards the axial direction by approximately 4°. A series of tests were required to establish the mainstream flow direction with and without the turbulence generator. First, the mainstream flow angle without the turbulence generator was measured directly just downstream of the second stage turning with a PIV system (TSI 2D system). These results were used to verify that the turning vanes turned the mainstream as designed, as well as to provide a relative point for measuring the effects of the turbulence grid on the flow angle.

Two additional PIV measurements were required to completely characterize the inlet flow incidence angle. The turbulence grid was placed in the test section and a second PIV measurement was taken just downstream of the turbulence grid. Once completed, the turbulence grid was removed and a third and final measurement was taken as the same position as the second measurement. These last two measurements of flow direction were used to determine the flow deflection the grid induced and subsequently quantify the inlet conditions for the high turbulence tests. The final incidence angles for the low- and high-turbulence cases were then used as the testing conditions for which airfoil pressure distributions were calculated using CFD.

2.3 Pressure Distribution

The 22 mid-span static pressure taps built into Test Article A were used to ensure the pressure distribution was set to the CFD predicted aerodynamic conditions. The two sets of radial taps, combined with taps at the same airfoil surface location at the mid-span, were used to ensure that the flow was aerodynamically 2-dimensional. This was important to the experimental intent as the test airfoil was a 2D extruded mid-span slice of an actual engine airfoil.

To measure the pressure distribution, the 22 mid-span static pressure taps were routed to 22 pressure transducers located outside the test section. A Pitot-static probe was located downstream of the airfoil model inside the test section to measure the downstream velocity. Once the Pitot-static probe was positioned properly, the 23 pressure measurements were collected and the coefficient of pressure for each pressure tap was calculated by:

\[
C_p = \frac{P_{\text{static, blade}} - P_{\text{total}}}{P_{\text{static, outlet}} - P_{\text{total}}} \tag{2}
\]

This process was repeated for radial measurements by rerouting 6 mid-span static pressure taps and the 12 off-mid-span static pressure taps to 18 pressure transducers. Coefficients of pressure were again generated using Equation 2.

Uncertainty for each \( C_p \) measurement was calculated by accounting for the uncertainty in the pressure measurements which was \( \delta p = \pm 2 \) Pa, which was primarily due to the
fossilized bias uncertainty inherent in the calibrations of the pressure transducers. Propagating these uncertainties according to procedures described in [12], the uncertainty for \( C_p \) was found to be \( \delta C_p = \pm 0.015 \).

### 2.4 Heat Transfer Coefficient

To measure the heat transfer coefficient, calibrated temperature measurements of the surface of the heat flux foil were taken with four IR cameras located at various positions around the airfoil model, shown in Figure 2. The locations of the cameras were set to maximize the viewing area of the airfoil. All IR cameras viewed the model through NaCl windows except for the far pressure-side camera, which viewed through a ZnSe window. Both NaCl and ZnSe are transmissive in the relevant IR wavelengths, but since their transmissivities vary, the same IR camera and window pairings were used throughout the series of calibrations and experiments. The mainstream temperature, mainstream velocity, current applied to the foil, and voltage drop across the foil were all measured. The local surface temperature measurements, \( T_w \), were then used to determine the local heat transfer coefficient, \( h \), with the following equation:

\[
h = \frac{q''_{\text{conv}}}{(T_w - T_{\infty})}
\]  

where \( q''_{\text{conv}} \) is the local convective heat flux. The local heat flux was determined using measurement of the total heat flux using the following equation:

\[
q''_{\text{totat}} = \frac{VI}{A}
\]

where \( V \) is voltage, \( I \) is current, and \( A \) is surface area of the heat flux foil.

Local values of heat flow from the heat foil due to conduction and radiation were determined as follows. To estimate conduction into the foam model, an internal surface temperature map of the airfoil model was generated using 10 internal surface thermocouples located at known positions inside the airfoil model. The internal surface temperature at each pixel location was then approximated with a cubic interpolation scheme [11]. This generated an external-internal temperature difference map to allow for the correction of the heat flux due to conduction through the walls (which typically comprised of ~5% of the heat flux) using the following equation:

\[
q''_{\text{cond}} = k \frac{\Delta T}{t}
\]

where \( t \) is the wall thickness.

The radiative flux losses from the model to the surroundings (which typically comprised of ~10% of the total heat flux) was calculated for each pixel as:

\[
q''_{\text{rad}} = \varepsilon \sigma (T_{\text{surf}}^4 - T_{\infty}^4)
\]

Accounting for these two heat flux processes, the convective surface heat flux was then calculated with:

\[
q''_{\text{conv}} = q''_{\text{totat}} - q''_{\text{cond}} - q''_{\text{rad}}
\]

Finally, the heat transfer coefficient map was generated using the convective heat flux using Equation 3. The average surface temperature range measured with the IR cameras was 325K-340K. This range was selected reduce the uncertainty in the calculated heat transfer coefficient.

Uncertainty in the heat transfer coefficient came from a number of sources, including uncertainty in the mainstream temperature (±0.5K), IR camera calibrations (±0.9K), radiation and conduction corrections (0.6%), and an additional lateral uncertainty due to some non-uniformity of an ideally laterally uniform temperature across the foil (±1.5%). The propagation of these uncertainties into the overall uncertainty for the heat transfer coefficient, \( h \), were determined using the procedure outlined in Kline and McClintock [12].

Local uncertainties over the width of the foil were averaged, and the lateral uncertainty was added to the total uncertainty when presenting lateral data. Uncertainty in \( h \) from all sources ranged from 5-10% of measurements.

3. CFD ANALYSIS

In order to assess the capability to predict \( h \) in the linear cascade, three turbulence models available in FLUENT 15.0 were used: 1) the 2-equation \( k \)-\( \omega \)-SST model with Low-Re corrections enabled, 2) the 3-equation Transition \( k \)-\( \omega \) model was utilized, and 3) the 4-equation Transition SST model. Each turbulence model was applied to the same 2D computational domain in FLUENT. All simulations also utilized the energy equation with a constant heat flux applied to the airfoil surface.

The computational domain used is shown in Figure 3. The computational domain included 1 axial chord length upstream of the model and 2 axial chord lengths downstream of the model. The mesh was an unstructured quad dominant mesh with a total of 68,000 nodes and 134,000 faces. A prism...
layer was used with \( y^+ = 1 \) for the grid coordinate nearest the wall. 2nd order discretization schemes were used for all variables. Constant pressure conditions were specified at the inlet and outlet, with a target mass flow rate set at the outlet to achieve the desired mainstream Reynolds number. The inlet static pressure was specified at atmospheric pressure which is very close to the experimental case (a low-speed recirculating wind tunnel); the outlet pressure is thus slightly lower than atmospheric due to the contraction of the area as it passes through the cascade. For each simulated case, the inlet incidence angle, turbulence intensity, and \( \lambda_f \) were all adjusted until they matched the corresponding experimental data seen in Table 2. The fluid properties used for the simulations are shown in Table 1.

![Figure 3. Computational Domain](image)

| Table 1. Fluid Properties Used for Simulations |
|-------------|-------------|
| Property    | Value       |
| \( \rho_{\infty} \) [kg/m\(^3\)] | 1.157       |
| \( \mu_{\infty} \) [kg/ms]   | 1.88 \times 10^{-5} |
| \( T_{\infty} \) [K]         | 305         |

In all simulated cases, the numerical values of \( Tu \) and \( \lambda_f \) were measured at ½ pitch in between the airfoils at the geometric leading edge so that \( Tu \) and \( \lambda_f \) were held constant for all simulations at this location. \( Tu \) is a parameter directly available in the software, and \( \lambda_f \) is calculated directly from \( k \) and \( \varepsilon \) at the measurement location. Converging on the proper \( Tu \) and \( \lambda_f \) magnitudes required iterating on the inlet \( Tu \) and \( \lambda_f \) for each simulation, since the inlet incidence angle and the distance from the domain inlet had an effect on the amount that \( Tu \) and \( \lambda_f \) changed from the specified inlet value. The iteration process was repeated for both \( Tu \) and \( \lambda_f \) until the values at the simulated measurement location were within 1% of the magnitude measured experimentally (i.e. \( Tu \) 5% ± 0.05% etc.).

4. RESULTS

In order to assess the capability to predict \( h \) in the linear cascade, four cases were tested and are summarized in Table 2. These cases consisted of high and low turbulence levels for the designed incidence angles of -25° and -35°. The turbulence integral length scale divided by axial chord length was measured to be \( \lambda_f/C = 0.06 \), with a precision uncertainty of ±2% chord length.

The upstream turning vanes were designed to turn the mainstream at angles of -25° and -35° relative to the axial direction, but as shown for the low \( Tu \) cases in Table 2, the actual turning angles were -25.0° and -33.8°. With the turbulence generator installed, the mainstream flow was turned towards the axial direction as it flowed through the array of turbulence producing rods. This resulted in a reduction in the turning angles to values of -21.2° and -30.1°. The turbine airfoil was designed to operate with an incidence angle of -30°, so experiments with high \( Tu \) were conducted with flow angle at essentially the design condition and with -8.7° offset from design conditions. The low \( Tu \) experiments also had similar range of incidence angles, but operated at +3.8° and -5.0° relative to the design flow angle.

<table>
<thead>
<tr>
<th>Table 2. Testing Conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nominal Case</td>
</tr>
<tr>
<td>----------------</td>
</tr>
<tr>
<td>-35° High ( Tu )</td>
</tr>
<tr>
<td>-25° High ( Tu )</td>
</tr>
<tr>
<td>-35° Low ( Tu )</td>
</tr>
<tr>
<td>25° Low ( Tu )</td>
</tr>
</tbody>
</table>

4.1 Pressure Distribution

As noted earlier, the test facility was designed to allow adjustment of the pressure distribution on the test airfoil to match computational predictions of the pressure distribution for an infinite cascade. An example of this matching is shown in Figure 4 for a section around the leading edge of the test airfoil. As is evident in this figure, \( C_p \) values at all 12 measurement positions were in a good agreement with the desired \( C_p \) distribution calculated computationally. This was true for all 22 measurement positions around the circumference at midspan (\( z/H = 0.5 \)) of the model airfoil for all test cases (full \( C_p \) profiles cannot be shown due to their proprietary nature).

![Figure 4. Example of Pressure Distribution Matching to CFD](image)

At six positions shown in Figure 4, x-1, 3, 5, 8, 10, and 12; surface static pressures were measured at three radial
Experimentally Determine External Heat Transfer Coefficient of a New Turbine Airfoil Design at Varying Incidence Angles

positions, \( z/H = 0.4, 0.5, \) and \( 0.6 \), to establish the radial uniformity of the pressure distribution. In most cases the radial variation in \( C_p \) was so small that the figure appears to show only one symbol, but there are three symbols that are superimposed on each other. This also serves as a confirmation of the accuracy of these measurements. In two cases, at \( x-10 \) and \( x-12 \), there was a very small radial variation of \( C_p \) measured approximately \( \pm 0.02 \), which was only slightly larger than the uncertainty of the measurement.

4.2 HTC Measurements

Contour plots of the measured distributions of \( h \) for all four cases tested are shown in Figure 5. Figure 5 also shows the locations of the stagnation lines initially determined from the CFD simulations and then later set experimentally in the wind tunnel. The heat transfer coefficient distributions were generally uniform in the radial \((z/H)\) direction. However, for the \(-25^\circ\) Low \( Tu \) case, there was a noticeable spanwise variation. This variation was determined to be due to small flow field non-uniformities that occurred with the \(-25^\circ\) turning vanes but not with the \(35^\circ\) turning vanes. Notice that with turbulence grid installed downstream of the turning vanes, i.e. the High \( Tu \) cases, the radial uniformity was much improved.

These plots also have the stagnation line for each incidence angle marked with a red dashed line. Note that \( S/S_{max} = 0 \) is the geometric leading edge point, and for all test cases the stagnation lines were towards the pressure side of the geometric leading edge, as expected. In each case the maximum \( h \) occurred close to the stagnation line, but slightly towards the suction side.

Direct comparisons of the \( h \) distributions with varying incidence angles are presented in terms of the laterally averaged \( h \), shown in Figure 6, along with the stagnation lines for each incidence angle. These results show very similar \( h \) distributions for the two incidence angles tested at each mainstream turbulence level. Given the measurement uncertainty of \( \delta h = \pm 2 \text{ W/m}^2\text{K} \), there was not a measurable effect of the incidence angle change. However, there was a consistent increase in \( h \) with the increase in mainstream turbulence, particularly at the leading edge. The increase in \( h \) at the stagnation line due to increasing the mainstream turbulence from \( Tu = 0.6\% \) to \( 5.0\% \) ranged from \( 11\% \) to \( 14\% \) depending on incidence angle. This is substantially less than the increase predicted by the correlation for leading edge heat transfer presented by Gandavarapu and Ames [13], e.g:

\[
Nu/Nu_0 = 1 + 0.04 \times TRL 
\]

where:

\[
TRL = Tu(D/fA_0)^{1/3}Re_0^{5/12}
\]

where \( D \) is the diameter of the leading edge. For the turbulence levels and length scales used in these experiments, this correlation predicts a \( 30\% \) increase in \( h \) at the leading edge. However, an elliptical leading edge model with a defined leading edge radius was used in the study by Gandavarapu and Ames [13]. Therefore, the asymmetry of the flow over an actual airfoil model, coupled with the fact that the minimum diameter of the airfoil model is not coincident with the stagnation line location or the locations of the strongest acceleration over the airfoil may have made the correlation less applicable to the present study.

Farther downstream from the leading edge on the suction and pressure sides of the model airfoil, the increase in freestream turbulence led to a \( 5\% \) increase in \( h \).

<table>
<thead>
<tr>
<th>Rig Inlet Angle</th>
<th>Stagnation Line ((S/S_{max}))</th>
<th>Peak Laterally Averaged HTC Location ((S/S_{max}))</th>
</tr>
</thead>
<tbody>
<tr>
<td>(-30.1^\circ)</td>
<td>(-0.058)</td>
<td>(-0.044)</td>
</tr>
<tr>
<td>(-21.23^\circ)</td>
<td>(-0.048)</td>
<td>(-0.046)</td>
</tr>
<tr>
<td>(-33.76^\circ)</td>
<td>(-0.067)</td>
<td>(-0.042)</td>
</tr>
<tr>
<td>(-25.03^\circ)</td>
<td>(-0.041)</td>
<td>(-0.038)</td>
</tr>
</tbody>
</table>

4.3 CFD Predictions of \( h \)

Figure 7 shows computational predictions with three turbulence models compared to the actual test data for the
-25° high \( Tu \) condition. In all three simulated cases, there is an under-prediction of \( h \) by as much as 15% at the leading edge, which is well outside the measurement uncertainty. The \( k \)-\( \omega \)-SST model predicts a premature laminar-to-turbulent transition on both the pressure-side and suction-side, with large over predictions of \( h \) thereafter. Both the \( k \)-\( \omega \) and the SST-Transition models predict the transition location farther downstream at a point beyond where the experimental measurements were taken. Furthermore, all models predict the heat transfer coefficient quite well for a very small portion of the pressure-side, although the \( k \)-\( \omega \) and the SST-Transition models are able to predict a much larger portion of the pressure-side, with the \( k \)-\( \omega \) model doing the best job. In addition to the pressure-side, the \( k \)-\( \omega \) and the SST-Transition models were able to predict overall trends on the suction-side of the airfoil after an \( S/S_{max} \) of 0.2. Although not presented here, the comparisons are similar for the -33.8° high \( Tu \) condition.

Figure 8 shows a comparison of the turbulence effects in the simulated and experimental -25° data sets for the \( k \)-\( \omega \) model only. In the leading edge region, the percentage enhancement of \( h \) due to turbulence is nearly the same in both the simulated and experimental cases. However, the enhancement of \( h \) downstream of the showerhead region is much more pronounced in the simulations with a nearly uniform increase in the simulated \( h \) distribution.

Figure 9 shows a comparison of the incidence angle effects in the simulated and experimental high \( Tu \) data sets, based on the \( k \)-\( \omega \) simulations only. Although the leading edge region is poorly predicted, as discussed previously, both the computational predictions and experiments show the effect of incidence angle on \( h \) is very slight even for the large incidence angle swings presented.

5. CONCLUSIONS

The results of this testing have been a milestone in verifying the functionality of a newly designed and built test section capable of testing a wide variety of both blade and vane turbine components at a range of potential incidence angles.

With the use of blockages and the adjustable outer wall of the test section, it was possible to set the pressure distribution very accurately relative to the CFD predicted distribution. This further ensured the capability of the wind tunnel to work for a variety of airfoil and vane designs.

There were unexpected results regarding the use of turbulence rods to generate the turbulence intensity required for testing. There is no correlation currently present in the literature for the incidence angle change due to a non-axially aligned turbulence grid, and it was presumed that the induced turning angle would be negligible since the turbulence grid had widely spaced rods. However, the turning of the flow was large enough to warrant direct measurement, and future work should look at adjusting the design to reduce such effects or increasing capability to predict the deflection, such as with CFD.
Measurements to determine the effects of a 9° shift of incidence angle on the heat transfer coefficient distribution around the airfoil showed that this airfoil design was insensitive to this change in incidence angle. This lack of sensitivity to the change in incidence angle is potentially due to the blunt airfoil geometry used in this study. Tests of the effects of increased mainstream turbulence showed a distinct increase in $h$ at the leading edge, but only a small increase farther downstream. The increase was less than predicted by a published correlation for stagnation point heat transfer enhancement by freestream turbulence. Furthermore, in each case the maximum $h$ occurred farther towards the suction side than the stagnation line. The effects seem to be due to a combination of a smaller airfoil radius, a stronger acceleration, and a higher local velocity at the location of maximum $h$ as opposed to stagnation line location.

Although the transition was predicted quite early in the SST-$k\omega$ model, the transition location was predicted very far downstream on the suction side for the other two models. It would be informative to measure the transition region HTC location and magnitudes and compare these to all of the simulated cases with a fifth IR camera in future tests.

CFD predicted the experimental $h$ for only some portions of the airfoil model within uncertainty. In general, the 3- and 4-equation models did a much better job at predicting $h$ than the 2-equation model. In the leading edge region, $h$ was underpredicted by all models, which could have occurred because of the types of turbulence models used, the shape of the airfoil, or many other reasons. Unfortunately, this means that even the 3- and 4-equation models inadequately predicted $h$ in the region of the airfoil that would experience the highest thermal loading without film-cooling. However, the percentage increase in $h$ in the leading edge region due to increased mainstream turbulence was well predicted.

ACKNOWLEDGMENTS

This work was sponsored in part by the Office of Naval Research under contract number N00014-09-D-0821/0006. The views contained herein are those of the authors and should not be interpreted as official policies of the Office of Naval Research, the U.S. Navy, or the U.S. Government.

REFERENCES


