**Effect of Nonuniform Inlet Conditions on Endwall Secondary Flows**

Exit combustor flow and thermal fields entering downstream stator vane passages in a gas turbine engine are highly nonuniform. These flow and thermal fields can significantly affect the development of the secondary flows in the turbine passages contributing to high platform heat transfer and large aerodynamic losses. The flow and thermal fields combine to give nonuniform total pressure profiles entering the turbine passage which, along with the airfoil geometry, dictate the secondary flow field. This paper presents an analysis of the effects of varying total pressure profiles in both the radial and combined radial and circumferential directions on the secondary flowfields in a first-stage stator vane. These inlet conditions used for the first vane simulations are based on the exit conditions predicted for a combustor. Prior to using the predictions, these CFD simulations were benchmarked against flowfield data measured in a large-scale, linear, turbine vane cascade. Good agreement occurred between the computational predictions and experimentally measured secondary flows. Analyses of the results for several different cases indicate variations in the secondary flow pattern from pitch to pitch, which attributes to the rationale as to why some airfoils quickly degrade while others remain intact over time. [DOI: 10.1115/1.1505849]

**Introduction**

Turbine inlet conditions in a gas turbine engine generally consist of temperature and velocity profiles that vary in the radial (spanwise for a linear cascade) and circumferential (pitchwise) directions resulting from combustor exit conditions. Depending on the conditions, these nonuniform profiles can have a strong influence on the nature of the secondary flows in the turbine platform region, also referred to as the endwall region. Secondary flows cause aerodynamic losses, high convective heat transfer, and make it difficult to film-cool the endwall region. These nonuniformities arise from combustor designs that contain film-cooling holes and/or slots to cool the liner and slots at the combustor-turbine interface. Several other factors contribute to the nonuniformities including the interaction of the liner cooling scheme, such as film-cooling holes, and the large dilution holes that are placed in the liner surface. There is also a dependency of the nonuniformities on the fuel and air mass flows in the combustor. Given that there is not always the same number of airfoils as dilution jets, this can cause a nonrepeating pattern for the turbine.

To determine the effects of combustor exit conditions on the secondary flowfields that develop in the passage of a nozzle guide vane, a progression of computations is presented in which increasingly more complicated effects were modeled. First, a uniform temperature with a two-dimensional boundary layer was simulated to compare with available experimental data. Second, several temperature and velocity profiles were simulated in which these profiles were assumed to vary only in the spanwise (radial) direction. The results of these studies indicated the importance of the incoming total pressure gradient. Third, total pressure profiles exiting the combustor (as predicted through CFD) were used as inlet profiles to the turbine vane for isothermal conditions. Clocking studies were also done to determine the effect of slightly shifting the profiles for this third case. The computations representing the first two cases were previously presented by Hermanson and Thole [1] but are briefly presented in this paper to provide a base-line understanding of these effects. All of the simulations were computed for a turbine vane geometry whereby the engine Reynolds number has been matched at low-speed conditions. Low-speed conditions were computed to allow direct comparison with measured flow fields for these same conditions in a large-scale wind tunnel simulation.

The following sections present a brief discussion of past studies, the CFD methodology and validation, the inlet profiles studied, and results of the study focusing on the relationship between inlet total pressure profiles and resulting secondary flows.

**Past Studies**

An increased understanding of the nature of endwall secondary flows has been and continues to be a major objective of research in the gas turbine industry. Numerous papers have been published discussing experimental and computational results in the endwall region. Reported results focus on heat transfer, secondary flow measurements, passage pressure loss measurements, and flow visualization, but there is only a small amount of documentation existing that analyzes these quantities in combination with the effect of spanwise and pitchwise gradients entering the first stage turbine vane from the combustor.

In 1947, Munk and Prim [2] showed that one would expect no changes in the streamline pattern and no distortion of a hot streak in the stator passage presuming a constant total pressure at the inlet to the stator. Much later, Lakshminarayana [3] investigated thermally driven secondary flows using a steady, inviscid theoretical analysis. The derived expressions indicated that gradients in total temperature will only generate secondary vorticity when there are accompanying radial gradients of entropy (total pressure) or Mach number. In the case of a constant Mach number, normal vorticity and total temperature gradients nullify each other. Although this condition is not realistic in a turbine engine, it is essential in understanding the control of secondary flows.

With the knowledge that there is some viscous-dominated region along the approaching platform to a turbine vane, a number of endwall secondary flow models were developed for an approaching turbulent boundary layer. Langston [4], Sharma and

---

Butler [5], Takeishi, et al. [6], and Goldstein and Spores [7] all proposed models and, although the models vary in complexity, each contains as a minimum:

• a leading edge horseshoe vortex formed from the inlet boundary layer which separates into the pressure-side and suction-side legs, and

• the passage vortex having the same sense of rotation as the pressure side leg of the horseshoe vortex developed due to the airfoil curvature.

The basic secondary flow models discussed generally assume a turbulent boundary layer profile along the approaching platform to the blade or vane. In realistic engine conditions, however, the combustor exit profile can contain large gradients in temperature and pressure in the spanwise and pitchwise directions. In fact, Barringer et al. [8] documented an experimentally simulated gas turbine combustor that showed wide variations in the pressure field that had little resemblance to the turbulent inlet boundary layer profile commonly assumed. Large temperature gradients created by film-cooling jets in the combustor liner were also reported.

While the experimental studies by Butler, et al. [9] and Shang et al. [10] have considered non-uniform inlet temperature profiles, the only experimental study to have considered whether there is an effect of a total pressure variation was that by Stabe et al. [11]. Stabe et al. simulated a liner flow through the use of a Combustor Exit Radial Temperature Simulator (CERTS). The CERTS used circumferential slots with no dilution holes. It was clearly identified from this study that changes did occur in the total pressure when using the CERTS as compared to not using the CERTS. Details are not available, however, for comparing the effects of the two different total pressure profiles on the secondary flowfield development and temperature distortion within the stator vane section.

Based on these studies, there is a clear need for an understanding of how total pressure variations at the inlet to a turbine vane section affect the development of the secondary flowfields. The work presented in this paper provides the first open literature study on the effects of radially and circumferentially varying inlet conditions on the secondary flow fields in a nozzle guide vane passage.

Turbine Vane Design and CFD Methodology. This section describes the turbine vane geometry that was studied and the methodology used to study the nonuniform inlet conditions to the turbine. While the profiles that varied in the radial direction alone were based on engine measurements, the computations for the radial/circumferential variations were performed in a two-step process. First, the combustor was simulated considering only the combustor geometry. Second, these combustor exit profiles were used as inlet boundary conditions for the turbine vane simulation. Note that low-speed conditions were simulated for the vane to allow for direct comparisons with simulations and experimental data previously acquired at nominally the same conditions except with a uniform flow field and a turbulent boundary layer on the approaching endwall.

Turbine Airfoil Description. The airfoil geometry used for these studies was a commercial first-stage stator vane, previously described by Radomsky and Thole [12,13] and Kang and Thole [14]. The characteristics of the vane geometry and flow conditions are summarized in Table 1. The vane is two-dimensional with the midspan cross-sectional geometry modeled along the entire span of the vane. The CFD simulations were computed for incompressible, viscous, low-speed conditions, thereby matching the Reynolds number but not the Mach number distribution. The effect of not matching the Mach number was addressed in Hermanson and Thole [15], which indicated similar secondary flowfield features between low and high Mach number cases. The primary effect of the Mach number is that the vane becomes slightly foreloaded for the low Mach number case and aftloaded for the high Mach number case. The effect of being foreloaded is such that the secondary flows are slightly stronger. For both the CFD analysis and wind tunnel experiments, the exit Reynolds number based on chord and exit velocity was .

The CFD simulations for the combustor were completed as described previously by Malecki, et al. [16]. The mesh generator used was the commercially available ICEM while the flow solver used was Allstar. Allstar is a proprietary derivative of the National Combustion Code ([17]) that is a pressure-based, finite volume, flow solver, which uses the conservative form of the discretized governing equations. Centered differencing was used with second order accuracy. A standard k-ε turbulence model with wall functions for near-wall modeling was used for the combustor simulations.

The CFD simulations for the turbine vane section were completed with a commercial software package by Fluent, Inc. [18]. Fluent is a pressure-based, incompressible flow solver for unstructured meshes. Second-order discretization was used for the Reynolds Averaged Navier Stokes (RANS) equations as well as the energy and turbulence equations. Fluent is especially applicable to three-dimensional endwall flows because the unstructured mesh capabilities allow the complex geometry of the stator vane to be modeled and because the code allows for solution-adaptive grids based on flow gradients to achieve grid-independent results.

The computational domain for the low speed simulations is depicted in Fig. 1. A domain representing one vane pitch was used to study grid independence, benchmarking of turbulence models, and effects of radial inlet profiles. Computations for three vane pitches were required to match the periodicity of the pitchwise
and spanwise varying inlet condition obtained from the combustor simulations. The inlet boundary condition was placed 75% of a true chord length (0.75C) upstream of the blade stagnation and the outlet boundary condition was placed 1.5C downstream along a line directed with the exit flow angle of the vane. Note that this location for the inlet boundary condition was also used as the location to set the combustor exit conditions as the inlet conditions for the turbine vane simulations. No additional dilution or cooling flows were applied downstream of the inlet boundary plane since the combustor simulations had already taken those effects into account. Since the combustor simulations did not include the effect of the vane on the flowfield, however, it was necessary to set the inlet conditions at 75% of a true chord upstream where the vane no longer affects the mean inviscid velocity.

Periodic boundary conditions were placed between the inlet, blade surface, and outlet. The vane was split across the pitch at the location of the dynamic stagnation point. The periodic boundary has been devised at the outlet of the domain to avoid highly skewed cells at the outlet/periodic surface intersection (shown in Fig. 1). A symmetry condition was applied at the vane midspan while a wall boundary condition was used to represent the endwall. The placement of the inlet and outlet boundary conditions was determined by two-dimensional CFD analyses of the midspan freestream flow.

The primary interest of this study was to discern the horseshoe vortex legs and the passage vortex convecting through the turbine vane passage. Figure 2 shows locations where the flow field predictions of the secondary flows will be compared in this paper. Note that these flow planes are placed normal to the mean flow direction (Vn). For clarity, the component of velocity, Vn, is zero at the midspan since this plane has a purely streamwise flow component.

Fig. 2 Locations and coordinates for benchmarking CFD with experimental measurements

Grid-independent was verified through a study using three different mesh sizes for a single periodic domain. For the coarsest mesh, consisting of $4.6 \times 10^5$ cells, the number of cells was conserved by placing the inlet and outflow boundary conditions closer to the vane than presented in Fig. 1 (at one-half chord upstream and one-half chord downstream) with a courser node spacing at the mid-plane. Two more refined grids were used in this study, with the inlet and outflow boundary conditions illustrated in Fig. 1 (0.75C upstream and 1.5C downstream), consisting of $8 \times 10^5$ cells and $1.3 \times 10^6$ cells.

As a check on the grid sensitivity, the average total pressure losses at several different positions through the cascade passage were calculated. Figure 3 compares the total pressure loss coefficients for all three mesh sizes. To compare the loss for each of the cases in this study the loss was calculated based on mass averaged values at the inlet. The total pressure loss results indicate that there are almost no differences between the two largest mesh sizes and an underprediction for the smallest mesh size. There is only a 0.66% difference between the two largest meshes at $X/C = 0.4$. For these studies the $8 \times 10^5$ cell mesh size was considered to be grid-independent. In addition, the secondary flow structures were compared to experiments for the various mesh sizes indicating sufficient grid resolution for secondary flow development predictions.

Prior to simulating various temperature and velocity profiles, the flow field predictions for the nozzle guide vane were validated through comparisons using laser Doppler velocimeter (LDV) measurements of Kang and Thole [14]. The standard k-ε (Laun-der and Spalding [19]) and RNG k-ε (Yakhot et al. [20]) turbulence models were benchmarked in the leading edge stagnation plane [1] showing that the RNG k-ε model provided the most accurate prediction of the horseshoe vortex roll-up from the models considered. Note that no changes were made to the recommended constants for any of the computations that were performed. The RNG k-ε model was expected to provide more accurate results since it contains additional terms in the transport equations for k and ε that are more suitable for stagnation flows and flows with high streamline curvature. The computational results of the RNG k-ε model were also compared to experimental data on six additional planes along the vane. Results for the Plane SS-1 (see Fig. 2) are presented in Figs. 4(a)-(b) with the experimental measurements to demonstrate the good agreement further.

Fig. 3 Grid sensitivity studies showing the total pressure loss through the passage

Journal of Turbomachinery

OCTOBER 2002, Vol. 124 / 625
downstream in the passage where both the passage vortex and suction side horseshoe vortex dominate near the endwall. The most notable difference seen in each of the planes is that the vortex is located slightly higher off the endwall, 1–3% of the span, in the CFD analysis as compared with the experiments. The reason for this difference may be attributed to deficiencies in the near-wall turbulence modeling.

Scaling Inlet Profiles to Low-Speed Conditions

Prior to computing the nozzle guide vane flows from both radially and circumferentially varying profiles (two-dimensional profiles), studies of radially (spanwise) variations alone (onedimensional) were performed to attain a more complete understanding of the physical nature of endwall secondary flows. By applying a combination of inlet conditions to the successfully benchmarked computational domain, the effects of velocity, temperature, pressure, and Mach number could be analyzed. A summary of these cases is given in Table 2. Note that all cases were computed for an inlet Reynolds number, Re_{inlet}, of approximately 2.4 x 10^5.

Profiles of velocity and temperature varying only in the radial direction previously presented in [1] are represented in cases 1–3 in Table 2 and shown in Figs. 5(a)–(b). The baseline case, case 1, directly modeled wind tunnel conditions, thus a 99% boundary layer thickness of 9.1% span was applied at the inlet. This 9.1% span boundary layer profile was generated using the 2-D boundary layer code, TEXSTAN (Crawford [21]) also for the turbulent quantities. In case 2, a temperature profile is applied in combination with the baseline boundary layer thickness to monitor effects of temperature gradient. A temperature gradient is applied with constant velocity (case 3). The temperature gradient is approximately linear from 0–32% span (endwall to freestream) based on experimentally measured engine pattern factor profiles (Kvasnak [22]). The temperature gradient used for cases 2 and 3 is similar to that used by Boyle and Giel [23]. The mass-averaged total temperature for each case is equal to the constant temperature of the baseline case.

The inlet profiles used in cases 4 and 5, shown in Figs. 6(a)–(c) and 7(a)–(c) simulate pitchwise and spanwise varying inlet conditions. Note that Y/P = 0, 1.5 and 3 are located at vane stagnation locations. Contours of normalized total pressure are shown at the inlet plane (Figs. 6(a), 7(a)) and the plane SP for each vane (Figs. 6(b), 7(b)). The local velocity normalized by the average value is shown in Figs. 6(c) and 7(c). Profiles of pressure, velocity, and
temperature were obtained from the combustor CFD simulations and used to determine the velocity profiles. Due to the nature of the coupled solver for highly compressible flows the total and static pressure profiles of the combustor at engine conditions could not be simulated simultaneously. Since the previous study concluded that the total pressure gradients approaching the vane are the driving factor for secondary flow development,

**Fig. 6** Inlet boundary conditions for case 4.

...scaled total pressure gradients from the computations of the engine combustor were simulated. The scaling was accomplished by calculating a velocity profile from the predicted total pressure and temperature profiles of the combustor exit assuming a constant static pressure (midspan value of engine simulations). The velocity profile was normalized by the average velocity exiting the combustor. This normalized profile, along with the average velocity required to match the inlet Reynolds number for the vane, was used to calculate the inlet conditions for the simulations. Since the inlet boundary condition for the CFD simulations also has a constant static pressure, the velocity profile imposed allowed for the total pressure gradients of the engine condition to be matched.

The resulting velocity profiles (note that the temperature was considered isothermal) were applied across three vane pitches with periodic boundary conditions applied outside of these three passages. The difference between cases 4 and 5 is simply a shift of the profiles that was 22% of one vane pitch. This shift provided a systematic comparison of the effect of the total pressure profiles. Two cases were considered so that nearly every combination of high and low velocity approaching the vane stagnation could be analyzed. A constant static temperature and pressure was applied in order to model a total pressure profile representing the characteristics of the jets. Although the profile at the combustor exit is not symmetric about the mid-span, the condition was assumed in the CFD simulations in order to achieve a reasonable size mesh of $2 \times 10^6$ cells.

**Effects of Radially Varying Total Pressure Profiles**

Prior to presenting results of the effect of both pitchwise/spanwise gradients on secondary flow development, a summary of cases 1–3 will be given. This is meant to provide the necessary baseline understanding on the effects of profiles varying normal to the endwall. For this purpose the focus will be on cases 1–3 summarized in Table 2.

Normalized total pressure contours at the leading edge, Plane SP (Figs. 8(a) and 9(a)) graphically display the relationship between stagnation pressure gradients and secondary flows further in the passage, Plane SS-1 (Figs. 8(b) and 9(b)). These secondary flow planes can be compared directly to those of case 1, which was previously shown in Fig. 4(a) for an isothermal flow with a boundary layer thickness that was 9% of the span. The total pressure gradient for the case 1 simply decreases towards the endwall because of the slower velocities in the boundary layer. The presence of a total pressure gradient such that the total pressure decreases as it approaches the endwall drives the flow towards the endwall. This results in the formation of the horseshoe/passage vortex for cases 1 and 2. For case 3, however, in which the total...
pressure is lower at the midspan, there is a reversed, counter-rotating vortex extending from the endwall to midspan. With a constant inlet Mach number (not shown in this paper), Hermanson and Thole [1] showed that no secondary flow vortex structures result since the total pressure is uniform. For the uniform Mach number case, there is, however a component of velocity sweeping from the pressure to suction sides of the vane.

Secondary flows near the endwall show only a slight difference from the baseline case even though there is a variation in the total temperature profiles (Figs. 4(a) and 8(b)). The real difference that occurs between these cases is further away from the endwall and towards the midspan where the flow actually forms another vortex. The larger counterrotating vortex predicted in cases 2–3 is a result of the temperature profile prescribed at the inlet which causes a negative gradient in total pressure from endwall to midspan. As seen in the streamwise vorticity contours, the temperature profile and resulting vortex has only a minimum effect on the streamwise vorticity relative to the vorticity generated near the wall.

Of particularly important interest in this parametric study are the distortions of the temperature profiles caused by the secondary flows. The temperature gradients exiting the combustor, sometimes referred to as hot streaks, are distorted by the secondary flow patterns in the first stage stator and create unsteadiness in the rotor downstream. These temperature profiles in combination with secondary flow pattern. For case 3 the endwall region experiences the highest temperatures across the plane. The cooler temperatures are eventually equal for each case, and the excess loss in $P_2$ cases 4 and 5 have regions of high and low pressure near the endwall and to about 15% span (shown in Figs. 6(a) and 7(e)). Due to the periodicity of the velocity profile each computational case simulated three vane pitches. The vane pitch for each case will be referred to by $P_1$, $P_2$, $P_3$, with $P_1$ corresponding to y/P = 0. The mass averaged total pressure loss through each of the six vane passages is shown in Fig. 10. These were calculated by averaging the inlet quantities across each pitch rather than averaging over the entire inlet so that a true comparison between pitches can be made. Figure 10 shows that the largest losses (by nominally 10% above $P_1$ and $P_3$) occurred in $P_2$ for both cases. By looking at the total pressure loss based on averaged values at the inlet, the loss coefficients at the entrance to the passage are approximately zero. Therefore, the loss associated with the inlet profile is essentially equal for each case, and the excess loss in $P_2$ (cases 4 and 5) must be due to secondary flows occurring in the passage.

Effects of Pitchwise and Spanwise Total Pressure Profiles

Two CFD simulations, cases 4 and 5, were analyzed to determine the effect of a combination of pitchwise and spanwise varying inlet conditions on endwall secondary flows. These cases consider a velocity profile with constant temperature in order to achieve an inlet total pressure profile representative of that of the cooling jets from the combustor. As presented in the previous section, the inlet total pressure is the driving factor in determining endwall flows. Although temperature distortions cannot be assessed using this method the pitchwise varying development of secondary flows can be evaluated. The total pressure profiles for case 4 and case 5 have regions of high and low pressure near the endwall to about 15% span (shown in Figs. 6(a) and 7(e)). Due to the periodicity of the velocity profile each computational case simulated three vane pitches. The vane pitch for each case will be referred to by $P_1$, $P_2$, $P_3$, with $P_1$ corresponding to y/P = 0. The mass averaged total pressure loss through each of the six vane passages is shown in Fig. 10. These were calculated by averaging the inlet quantities across each pitch rather than averaging over the entire inlet so that a true comparison between pitches can be made. Figure 10 shows that the largest losses (by nominally 10% above $P_1$ and $P_3$) occurred in $P_2$ for both cases. By looking at the total pressure loss based on averaged values at the inlet, the loss coefficients at the entrance to the passage are approximately zero. Therefore, the loss associated with the inlet profile is essentially equal for each case, and the excess loss in $P_2$ (cases 4 and 5) must be due to secondary flows occurring in the passage.
The flowfield and streamwise velocity on plane SS-1 for each of the passages in case 4 is presented in Figs. 11a–c with the streamline patterns as shown in Fig. 11d, while case 5 is shown in Figs. 12a–d. The flowfields in each of the six passages exhibit a unique flow pattern demonstrating the effect of the pitchwise varying gradients. Elements of the flowfields presented in the spanwise varying parametric study can be detected. Both the magnitude and direction of the endwall vortices are very different even between neighboring pitches.

The flowfield in P1 of case 4 reveals a similar pattern to case 3. Flow is moving up the pressure surface of the vane and there is a very strong $V_{ns}$ component toward the suction side at about 30% span. There is no sign of the pressure side horseshoe vortex (PSH) or suction side horseshoe vortex (SSH). In the next passage, P2, the flowfield does exhibit the passage vortex as seen in case 1. This vortex occurs further off the endwall, further from the pressure surface, and occupies a larger region of the endwall than previously documented, however the flow down the pressure surface and across the endwall indicates the presence of the passage vortex. In this case, there is no SSH near the suction surface, and the flow across the endwall continues to encounter the suction surface. Passage P3 again shows a flowfield with similarities to case 3 with flow moving towards the midspan along the pressure surface. This counter rotating vortex occupies a greater area of the passage than is shown in P1. The streamwise velocity contours show that the largest amounts of skewing occur near the vane endwall for the conditions where the cross-flow velocity component exists.

Although the inlet profile was shifted by only $Y/P=0.22$ between cases 4 and 5, very different flowfields are shown in Figures 12a–d from what was seen in Figs. 11a–c. P1 (Fig. 12a)) has flow moving up the pressure surface to form a large vortex similar to P1 in case 4, but this time the SSH is present in the suction side-endwall juncture. A flowfield representing a passage vortex is again seen in P2 (Fig. 12b)). This vortex occurs closer to the endwall and pressure surface showing more similarity to case 1. There is no SSH revealed in the vector plot for P2. There are only very small magnitudes of secondary flows on P3, but the pattern, which does exist is possibly the most complex demonstrated on any of the passages. A vortex resembling the passage vortex is present near the suction surface up to about 15% span. On the pressure surface, also at about 15% span, flow is directed toward the midspan in the opposite direction of the vortex near the endwall. Again, for these simulations the streamwise velocity shows that the streamwise flow is affected most in P2 due to the large secondary flows.

In order to explain the flow phenomena depicted in Figs. 11 and 12...
the gradients in total pressure in the leading edge stagnation plane for each passage again must be examined. In this analysis the gradients in total pressure across the pitch are just as important and must also be the focus. Recall that the static pressures at the inlet are approximately constant for these cases. Figures 6 and 7 present the normalized pitchwise total pressure gradients \( a \) with the normalized spanwise gradients in Plane SP \( b \) for P1, P2, and P3 for cases 4 and 5.

The locations of the stagnation planes are indicated in Figs. 6\( a \) and 7\( a \) by dashed lines. Arrows indicate the tendency of flow to move from regions or pockets of higher pressure to lower pressure. The high-pressure gradients located at \( y/P = 0.8 \) would be expected to drive flow in the pitchwise direction towards the lower pressure. Similarly, flow from 15% span should have a strong component of spanwise velocity towards the endwall. Much smaller magnitudes of secondary flows are expected at \( y/P = 0 \) due to only small pressure gradients. This plot shows that the flow will already have a 3-D nature as it moves 75% of a chord length from the inlet boundary to the vane stagnation with a very thin boundary layer developing along the endwall. These gradients will also become more distorted as they are accelerated inside the vane passage.

Spanwise gradients for case 4, Fig. 6, also indicate (by arrows) the favorable direction of flow due to the total pressure profile. Both P1 and P3 have profiles such that the flow may divide at 20% span and move up or down the vane. The movement here will also depend on the pitchwise distortions, which move the flow along the out-of-plane direction of the plot. Flow should move up the vane surface at P3. Note that the magnitude of the pressure gradients in the spanwise direction are quite small relative to the pitchwise gradients. The spanwise gradients are also small relative to the inlet conditions of the spanwise varying inlet profiles (cases 1–3).

The pitchwise total pressure gradients for case 5 are again marked with vane spacing and predicted flow in Fig. 7. Since this pressure profile is the same as in Fig. 6 with a 22% shift, the same basic motions can be expected as flow approaches the vane. Once the vane is encountered, however, the interaction with the stagnation and pressure/suction surfaces creates different flowfields from case 4.

The flowfields can be further predicted for case 5 by looking at the pressure gradients in plane SP, Fig. 7. The profile at P1 should drive flow up the vane surface to at least 30% span where an adverse pressure gradient is then encountered. The spanwise pressure gradients in P2 greatly resemble a boundary layer profile, such as that in case 1. The gradients occur up to almost 20% span in this case, but with a smaller pressure deficit than shown for case 1 (Fig. 7). The profile of P3 should also direct flow towards the midspan at the leading edge.

Conclusions

Prior to studying the effects of inlet conditions on secondary flows, CFD results from a baseline case were benchmarked against the flowfield measurement indicating good agreement. The effects of both purely radial and combined radial/circumferential inlet profiles on the development of secondary flows were determined from a parametric study. Four cases with various combinations of inlet velocity and temperature profiles were compared with the baseline case to determine the effect of spanwise temperature gradients. Application of a temperature gradient in combination with an inlet boundary layer reduced the streamwise vorticity and spanwise velocity associated with the secondary flows. Flow also moved up the vane surface forming a large counterrotating vortex above the boundary layer to the midspan. A temperature gradient applied at the inlet with a constant velocity resulted in large vortex rotating in the opposite direction as the passage vortex with flow moving away from the endwall up the vane surface. This vortex had very little streamwise vorticity. A constant Mach number at the inlet showed no vortex formation. The total pressure gradient is the driving factor for determining the magnitude of secondary flows in the passage.

Temperature contours showed that the secondary flow pattern distorts the temperature for each case. With a temperature gradient and a turbulent inlet boundary layer there was a migration of the cooler temperature to the suction surface. The cooler temperatures moved up the pressure surface in the case of a temperature profile with a constant inlet velocity. Since there were no secondary flows with a constant inlet Mach number there was also very little temperature distortion.

Secondary flow patterns were predicted for two cases with a pitchwise and spanwise total pressure gradient. Six very unique flow patterns resulted from the inlet profile. Characteristics of the flows could be compared with and predicted from the cases analyzed that considered only spanwise inlet profiles. In one passage, flow moved up the vane surface while in the neighboring passage the classic flow pattern depicted in flow models was formed. Again, the flow patterns were a function of the total pressure gradients in the leading edge region in both the spanwise and pitchwise directions. In regions of low spanwise pressure gradient at the leading edge there was very little streamwise vorticity downstream in the passage.

The results from the pitchwise and spanwise varying profiles have not been previously documented. The results demonstrate the need to consider realistic combustor exit profiles in stator design. It may be possible to align high and low pressure regions exiting the combustor with the first stage stator to minimize the resulting secondary flows. The presented results also provide reasoning for gas turbine field experience revealing that some airfoils show heavy degradation leading to a shorter life while neighboring components remain intact over time.

Acknowledgments

The authors also acknowledge that this work was completed at the University of Wisconsin where K. Hermanson completed her MS degree.

Nomenclature

\[ C = \text{true chord of stator vane} \]
\[ k = \text{turbulent kinetic energy} \]
\[ Ma = \text{Mach number} \]
\[ m = \text{mass flow through passage} \]
\[ n = \text{coordinate normal to inviscid streamline} \]
\[ p = \text{static pressure} \]
\[ P = \text{pitch} \]
\[ P_o = \text{total pressure} \]
\[ Re = \text{Reynolds number, } Cu/V \]
\[ v = \text{coordinate aligned with inviscid streamline} \]
\[ S = \text{span of stator vane} \]
\[ T_s = \text{static temperature} \]
\[ U = \text{freestream velocity} \]
\[ U, V, W = \text{absolute velocity components} \]
\[ u, v, w = \text{secondary flow plane, transformed velocity components} \]
\[ V_s = \text{streamwise velocity, } u \cos \psi_{ms} + v \sin \psi_{ms} \]
\[ V_n = \text{normal velocity, } -u \sin \psi_{ms} + v \cos \psi_{ms} \]
\[ V_v = \text{spanwise velocity, } w \]
\[ X, Y, Z = \text{absolute, stationary, coordinate system} \]
\[ x = \text{distance normal to secondary flow plane} \]
\[ y = \text{distance tangent to secondary flow plane} \]
\[ Y_s = \frac{\text{pressure loss coefficient}}{2(\rho \mid v_{\text{inlet}} - P_r / \rho U^2_{\text{inlet}}) dA / \dot{m}} \]

\[ \delta = \text{boundary layer thickness} \]

\[ \Delta P = \text{normalized pressure} \]

\[ \varepsilon = \text{dissipation} \]

\[ \nu = \text{viscosity} \]

\[ \rho = \text{density} \]

\[ \Omega_x = \text{streamwise vorticity}, \quad \Omega_y = x\text{-vorticity}, \quad \Omega_z = y\text{-vorticity}, \]

\[ \psi_{\text{ms}} = \text{midspan turning angle, } \tan^{-1}(v_{\text{ms}}/u_{\text{ms}}) \]

**Subscripts**

\( \text{inlet} \) = value at 0.7 C upstream of vane

\( \text{max} \) = maximum value in profile

\( \text{mid} \) = value at vane midspan

\( \text{ex} \) = value at vane exit

\( \text{avg} \) = mass-averaged value

**References**


