Leading-Edge Film-Cooling Physics—Part III: Diffused Hole Effectiveness

William D. York
James H. Leylek
Department of Mechanical Engineering, Clemson University, Clemson, SC 29634

A proven computational methodology was applied to investigate film cooling from diffused holes on the simulated leading edge of a turbine airfoil. The short film-hole diffuser section was conical in shape with a shallow half-angle, and was joined to a plenum by a cylindrical metering section. The diffusion resulted in a film-hole breakout area of 2.5 times that of a cylindrical hole. In the present paper, predictions of adiabatic effectiveness for the cases with diffused holes are compared to results for standard cylindrical holes, and performance is analyzed in the context of extensive flowfield data. The leading edge surface was elliptic in shape to accurately model a turbine airfoil. The geometry consisted of one row of holes centered on the stagnation line, and two additional rows located 3.5 hole (metering section) diameters downstream on either side of the stagnation line. Film holes in the downstream rows were centered laterally between holes in the stagnation row. All holes were angled at 20 deg with the leading edge surface, and were turned 90 deg with respect to the streamwise direction (radial injection). The average blowing ratio was varied from 1.0 to 2.5, and the coolant-to-mainstream density ratio was equal to 1.8. The steady Reynolds-averaged Navier-Stokes equations were solved with a pressure-correction algorithm on an unstructured, multi-block grid containing 4.6 million finite-volumes. A realizable k-ε turbulence model was employed to close the equations. Convergence and grid-independence was verified using strict criteria. Based on the laterally averaged effectiveness over the leading edge, the diffused holes showed a marked advantage over standard holes through the range of blowing ratios. However, ingestion of hot crossflow and thermal diffusion into the second row of film holes was observed to cause significant, and potentially detrimental, heating of the film-hole walls.

[DOI: 10.1115/1.1559899]

1 Introduction

To achieve high cycle efficiency in modern gas turbine engines, the temperature of combustion gases entering the turbine is typically at a temperature greater than the metal airfoils could withstand without a carefully designed cooling scheme. One of the most effective and often used methods for cooling the airfoils is film cooling. Air is bled from the compressor, sent directly to channels inside the blades and vanes, and then injected to the exterior surface through an array of small holes. Film cooling is especially critical at the leading edge of the turbine blades, since the largest thermal load occurs here and the cooling flow effects the aerodynamics and heat transfer of the entire airfoil. The problem is complicated by difficult mechanisms, including a stagnating freestream, strong acceleration and curvature, and interactions between the required multiple rows of coolant jets.

The gas turbine industry currently relies on a database of one and two-dimensional empirical correlations and engine performance data to design film-cooling schemes. A three-dimensional, predictive tool could potentially revolutionize cooling design and provide quick turn-around and relatively low cost. Computational fluid dynamics (CFD) can fill this role, but only if it is proven to give consistently accurate results. Toward that goal, the present study applies a systematic computational methodology to the leading-edge film-cooling problem. Validation of the methodology in the prediction of adiabatic effectiveness was achieved in Part I of this series of papers by York and Leylek [1].

This portion of the research program is an excellent case study in using CFD effectively in the actual design process. Currently, film-cooling holes on the leading edge of turbine blades are almost exclusively cylindrical in shape. Numerous numerical and experimental research has shown that exit-diffused film holes can provide far better cooling performance over cylindrical holes when used in the suction and pressure-side hole rows. The shaped hole may also provide increased performance on the leading edge, but little data is available to the designer on this configuration. On the request of an industry sponsor, film holes with a conical diffuser at their exits were simulated using the computational methodology, which was previously validated with experimental data on the standard holes. The sponsor received fast and reliable information on the conical diffuser hole design to help determine feasibility for use in current engine programs.

2 Literature Review

In the past two decades, there has been a fair amount of research, both experimental and numerical, on the leading-edge film-cooling problem. Notable experimental studies include the work of Mick and Mayle [2], Mehendale and Han [3], Ekkad et al. [4], and Salcudean et al. [5]. These researchers measured the adiabatic effectiveness and heat transfer coefficients on film-cooled leading edge models. All of these authors employed cylindrical holes in their tests. A more detailed explanation of the papers listed in the foregoing may be found in Part I of this series [1].

Cruse [6] used an infrared imaging technique to acquire detailed adiabatic effectiveness data on a film-cooled leading edge that was elliptical in shape, similar to an actual turbine airfoil. Again, the film holes were cylindrical in shape. The conditions of Cruse [6] were simulated in Part I of the present set of papers [1], and the computations were validated against this data.

The only known study in the open literature that examines the use of shaped holes on the leading edge is that of Reiss and Böles...
The authors used a transient liquid crystal technique to make detailed measurements of the adiabatic effectiveness and heat transfer coefficient on a cylindrical leading edge. Five staggered rows of cooling holes were located at 0, ±20, and ±40 deg with respect to the stagnation line. In addition to standard cylindrical holes, the authors investigated holes with exits expanded in the flow direction and in the lateral direction. The holes with the forward expansion performed the best, generally exhibiting the highest spanwise averaged effectiveness and lowest heat transfer at all blowing ratios. The laterally diffused holes gave a small improvement in effectiveness over the standard cylindrical holes, but showed large increases in heat transfer downstream of the holes at high blowing ratios.

On the numerical side, a blind test was organized to assess the ability of CFD in predicting leading edge film cooling. Chernobrovkin and Lakshminarayana [8], Lin et al. [9], Martin and Thole [10], and Thakur et al. [11] each ran simulations following the experimental conditions of Cruse [6] with a cylindrical leading edge model. Only a single blowing ratio was examined (M = 2.0). The different authors applied varied grid types, discretization schemes, and turbulence models to the problem. The agreement with the experimental effectiveness data was mixed. There are no known numerical studies of shaped or diffused holes on the leading edge in the literature. The present paper aims to add significantly to the knowledge base in this area by providing surface data for conical diffuser holes and explaining the physics involved.

3 Methodology

3.1 Computational Model. The computational model employed in the present work is identical to that described in Part I [1], with the only exception being the addition of a diffuser section to the film holes. The numerical domain is shown in Fig. 1. The model developed and validated in Part I [1] was derived from the experimental setup of Cruse [6]. The leading edge was elliptical in shape, with a major axis of 132 mm and a minor axis of 66.0 mm, and the thickness of the leading-edge wall (separating the freestream from the plenum) was 25.4 mm. A symmetry plane was used to impose the freestream stagnation condition and was normal to the elliptic leading edge at it widest part. One row of cooling holes was located along the stagnation line (Row1), and a second row of holes was located 3.5D downstream (Row2) and was staggered such that holes in this row were centered laterally between holes in the Row1. All film holes had an injection angle of α = 20 deg and a compound angle of φ = 90 deg (spanwise injection). The holes had a lateral spacing (within each row) of S = 7.64D between their centerlines.

The conical diffuser (CDIFF) hole geometry was specified by a gas turbine engine manufacturer from design considerations, and is shown schematically in Fig. 2 with a comparison to the standard cylindrical hole. The CDIFF geometry would result from driving a cone-shaped cutting tool into the end of the cylindrical film hole, such that expansion is identical in all directions. The diffuser section had a shallow half-angle and it began only a short distance upstream of the exit plane (on the hole centerline). This resulted in an area ratio of AR = 2.5, meaning that the CDIFF breakout area was 2.5 times greater than the standard (REF) holes. The cylindrical “metering” section at the upstream side of the holes had a diameter of D = 6.32 mm to match the REF holes, as did the total hole length of 11.75D.

Periodic planes served as the lateral bounds of the domain and were located halfway between the midpoints of two adjacent holes (in different rows), and were spaced apart by the film-hole pitch (5). Because of the significantly wider breakout of the CDIFF holes, the periodic plane had to pass through the Row1 diffuser section, and thus the Row1 hole breakout is actually split with a portion located on each side of the computational domain, as seen in Fig. 1. When the periodic and the symmetry conditions are imposed, the domain simulates a leading edge with three rows of cooling holes and infinite span that is identical to the configuration with the REF holes in Part I [1].

Mainstream air at a temperature of 300 K entered the domain through a uniform velocity inlet with U∞ = 10 m/s located 140 mm upstream of the stagnation point. The turbulence intensity at the mainstream inlet was set at 0.2%, and the length scale of turbulence was specified as one-tenth of the inlet height to match the simulations from Part I [1]. Coolant air entered the plenum through a uniform velocity inlet at a temperature of 166.5 K, which produced a density ratio of DR = 1.8. The plenum inlet velocity was varied to achieve specific blowing ratios. The blowing ratios discussed in this study are calculated from the mean velocity in the cylindrical metering section averaged over all holes. Thus, for a specific blowing ratio, the CDIFF and REF cases have the same overall mass flux of coolant. All fluid exited the domain well downstream of the leading-edge model through a convection outlet. A flat wall joined the curved leading edge surface to the outlet, and the upper boundary of the domain was a solid wall. All walls were specified as adiabatic (zero heat transfer).

3.2 Grid. To allow a fair comparison with the validated results for the standard holes from Part I [1], the finite-volume mesh employed was virtually identical to that used the REF simulations.
A superblock, unstructured approach was used to maximize overall grid quality, and the mesh was comprised of hexahedra, triangular prism, and tetrahedra cells. To fully resolve the viscous sublayers, all cells along walls were sized such that the first grid point was located at a maximum wall distance of $y^+ = 1$. Due to the larger volume of the film holes with the diffuser, the CDIFF cases contained 4.8 million total cells, approximately 30% more cells than the REF cases. A view of the surface mesh on the leading edge and film-hole wall at the Row2 breakout is shown in Fig. 3. Grid generation was accomplished with GAMBIT and T-Grid software from Fluent, Inc.

3.3 Simulation Details. The steady Reynolds-averaged Navier-Stokes equations were solved with the aid of a pressure-correction algorithm and a multi-grid convergence accelerator in Fluent 5 software from Fluent, Inc. Discretization of the governing equations was second-order accurate. The realizabile $k$-$\varepsilon$ (RKE) turbulence model of Shih et al. [12] was used to close the set of equations. The satisfaction of the realizability constraints in the $k$-$\varepsilon$ model eliminates the spurious production of turbulent kinetic energy in regions of large fluid strain. The RKE model is therefore expected to give superior results to the standard $k$-$\varepsilon$ model for the flowfield of the present study, while retaining the economic advantage of the two-equation model. A two-layer near wall treatment, in which the $k$-equation was solved in the wall-adjacent region, was used to resolve the turbulent boundary layers down to the walls.

The solution was declared as converged only when strict criteria were satisfied. First, residuals of the governing equations, normalized by their respective inlet fluxes, dropped below 0.1%. Second, the global mass and energy imbalances dropped below 0.01%. Lastly, profiles of flowfield variables ($U$, $T$, $P$, $k$, and $\varepsilon$) and surface temperature showed negligible change with further iterations. To verify grid-independence, the background mesh was refined in regions of large gradients of the flowfield variables, such that there was a 20% increase in the total number of cells. A hanging-node adaption technique was employed to retain the high quality of the original grid. Due to the extremely high density of the background grid, changes of less than 2% for all flowfield variables and the adiabatic effectiveness were observed when switching to the refined grid. The simulation was therefore declared to be grid-independent with the original grid. The simulations were run on 20 parallel processors on a Sun Microsystems Ultra 6500 computer with 10 gigabytes of RAM. A single iteration took approximately 40 s, and about 3000 to 4000 iterations were required for convergence at each simulated blowing ratio.

4 Results and Discussion

This section details the performance of the conical diffuser holes by first examining the adiabatic effectiveness and then looking at the flowfield characteristics. Again, note that since the blowing ratio is defined with the velocity in the metering section, the CDIFF holes have the same coolant mass flow rate as the REF holes at a specific value of $M$. The basic principle behind the CDIFF hole is simple. As the cross-sectional area is gradually increased toward the film-hole exit plane (FHEP), the average velocity of the coolant decreases due to the diffusion. The breakout area of the CDIFF hole is 2.5 times greater than the REF breakout area. Velocity is inversely proportional to the area, so the average velocity across the FHEP is 40% smaller for the CDIFF hole. If the diffuser works well and a uniform velocity exists over the FHEP, the momentum of the coolant will be only 16% of the value in the REF case, since momentum varies with the square of the velocity. Theoretically, the reduction in coolant $y$-momentum should keep it closer to the surface, giving better thermal protection of the leading edge as compared to film holes without diffusers.

4.1 Surface Results. Contours of adiabatic effectiveness on the “unwrapped” leading edge with CDIFF holes for the blowing ratios $M = 1.0$ and $M = 2.0$ are given in Fig. 4. At the low blowing ratio, a region of high effectiveness trails directly aft of the Row1 hole. Because of the reduced momentum due to the diffuser, the coolant does not travel laterally and therefore does not protect the region between holes on the stagnation line. Another area of relatively high effectiveness is seen to lie directly downstream of the Row2 hole, and again the lack of lateral motion of the coolant is observed. At the higher blowing ratio of $M = 2.0$, a region of effectiveness approaching unity lies just aft and to the TE side on the Row1 hole. The coolant generally protects the stagnation line, before traveling downstream and directly into the Row2 jet. A fairly wide region of elevated effectiveness is seen aft of Row2. The coolant footprint is significantly wider than for the REF holes, for which contours are given in Part I of the paper [1]. While there is still a “hot streak” for the CDIFF holes aft of Row2, this unprotected region is far reduced in size as compared to the REF case.

Plots of the laterally averaged effectiveness versus streamwise distance from the stagnation line are given in Fig. 5, along with the curves from the REF simulations. At the lowest blowing ratio of $M = 1.0$, the effectiveness was observed to be zero over most of the area between Row1 holes (Fig. 4(a)). This translates to a very low laterally averaged effectiveness of $\bar{\eta} = 0.2$ at the stagnation line (Fig. 5(a)). At the downstream edge of Row1, $\bar{\eta}$ increases to a local maximum of about 0.55, which is over 20% greater than the REF case. With decent coolant coverage between holes in Row2, there is a spike in $\bar{\eta}$ at the streamwise location of Row2 ($x/D = 3.5$). The wide coolant footprint aft of Row2 results in a nearly constant 20% advantage in $\bar{\eta}$ over the REF case on the remaining portion of the leading edge surface.

As $M$ increases, the effectiveness increases in region between adjacent holes in Row1, and at $M = 2.0$, the value of $\bar{\eta}$ at the stagnation line reaches the REF value of 0.8. For all blowing ratios, the CDIFF holes give values of $\bar{\eta}$ that are 0.05 to 0.10 greater than the REF holes in the region between the two rows. At the blowing ratio $M = 1.5$ and above, the Row1 coolant flows over the Row2 hole and merges completely with this jet. This leaves most of the area between the Row2 holes unprotected, causing the laterally averaged effectiveness to significantly fall at $x/D = 3.5$. The most severe case is at $M = 2.0$, where the Row1 jet flows directly over the Row2 hole, and the local value of $\bar{\eta}$ drops from 0.4 to under 0.2 (Fig. 5(c)).
Also, as the blowing ratio increases, the region of high effectiveness aft of Row2 decreases in width and migrates from the LE side of the hole to the TE side. For all $M$, however, the coolant footprint downstream of the Row2 hole is spread over a greater lateral distance than in the REF case. Because of this behavior, $\bar{\eta}$ is up to 40% greater downstream of Row2 for the CDIFF holes. Overall, in terms of surface results, the CDIFF holes perform superior to the standard cylindrical holes. With a couple of exceptions at the location of a row of holes, the conical diffuser holes give a significantly higher laterally averaged effectiveness over the entire leading edge through the range of blowing ratios.

### 4.2 Flowfield Physics

This section analyzes the leading edge film-cooling flowfield with the conical diffuser holes in place. Comparisons are made to the REF cases, which are described in Part I of this series [1]. Since the blowing ratio was defined with the average coolant velocity in the long cylindrical metering section, which has the same diameter as the REF holes, coolant flow rates through the two hole geometries are equal. Thus, the CDIFF case exhibits the same fluid mechanics as found in the REF configuration from the film-hole inlet to the beginning of the short diffuser section. The coolant was observed to enter the diffuser section with a nearly uniform velocity profile and negligible secondary flow.

Because of a nearly uniform pressure field along the stagnation line and over the Row1 FHEP, the diffuser works well for this hole row. Figure 6 shows contours of the normalized $y$-velocity at the FHEP of the Row1 hole. The velocity is nearly constant across this plane, with $V_y$ values of 0.4 to 0.5 (40 to 50% of the value in the REF case). Because of the small normal component of momentum, the Row1 jet has a much shallower trajectory than the REF case. This is seen in contours of normalized temperature on three planes of constant $z$-coordinate in Fig. 7. In the first two planes to the right of the hole centerline, the contour lines are much more flat and spread out in the streamwise direction than in the REF case. Since the coolant does not have as much lateral momentum, it is turned in the streamwise direction quickly. By the third plane (at $z/D = 5.7$), the low temperature contours at the surface have disappeared as the coolant has turned downstream already. Though the coolant does not travel as far laterally as the REF case, the expanded breakout of the diffusers greatly reduce the metal distance between adjacent film holes and coverage along the stagnation line is excellent.

At Row2, the flow in the diffuser section is strongly influenced by the interaction of the crossflow with the emerging jet. The physics are largely the same as in the REF case. A high-pressure region develops at the upstream half of the hole where the crossflow slows as it meets the dense coolant jet. As a result, coolant migrates toward the downstream edge of the diffuser section. The development of this jetting region is clearly seen in the case of $M = 2.0$ in Fig. 8, lines on constant normalized $y$-velocity on a plane cutting through the Row2 film hole from upstream edge to downstream edge. Views of the normalized velocity contours on the Row2 FHEP for $M = 1.0$ and 2.0 are given in Fig. 9. The highly non-uniform momentum across the exit plane is evident. At the low blowing ratio, the full upstream half of the hole breakout is occupied by a region of zero normal velocity, meaning that essentially no coolant leaves the hole on this side. In fact, there is an area near the upstream edge where the value of $V_y$ is actually negative, the implication of which are discussed later. Velocities rapidly increase toward the downstream edge of the hole near the LE point, where $V_y$ exceeds a value of 3.0. At $M = 2.0$, a similar pattern develops, although the jetting is not as severe, with a maximum $V_y$ of near 2.1 along the downstream edge. This maximum value of the velocity is slightly less than in the REF case (where $V_y$ was locally above 2.7) since the coolant jetting region extends across the significantly wider breakout of the diffuser section. The diffuser does not work optimally for Row2, but the momentum of the coolant is reduced slightly from the REF case as it exits the hole.

In Fig. 10, contours of normalized temperature are displayed on three planes of constant $x$-coordinate downstream of the Row2 hole. The dashed lines in this figure bound the region on the leading-edge surface that sees a temperature of less than or equal to $\theta = 0.3$. In At $x/D = 4.5$, just downstream of the edge of the diffuser breakout, the jet core is observed to be spread out over a distance roughly equal to the lateral dimension of the FHEP (LE to TE). As in the REF simulation, the coolant has penetrated further into the mainstream on the LE side of the hole (right side in this view). However, due to the smaller velocity in the jetting region along the downstream film-hole wall, the coolant does not lift off the surface as far as in the REF case. A fairly strong streamwise vortex lies near aft of the LE side of the hole, as seen in Fig. 11, in-plane velocity vectors at $x/D = 4.5$. But since coolant lift-off is not as great for the CDIFF hole, the vortex does not bring hot crossflow under the jet core to the extent of the REF case. Secondary flow is clearly visible in the velocity vectors in this plane. Despite some movement of hot crossflow underneath the coolant jet core on the right side, a large portion of the jet is in contact with the surface on the left side, and the CDIFF hole gives
better lateral coverage than the REF hole. This trend continues downstream, and at \( x/D = 6.5 \) and \( x/D = 8.5 \) the coolant is located closer to the surface and is exhibits a greater lateral spread than the REF case (Fig. 10).

Interaction between the two rows of coolant jets from the CDIFF holes is impossible to avoid since the breakout planes of the film holes are so wide. At \( M = 2.0 \), the Row1 coolant, which has less lateral momentum than the REF case, travels directly over the Row2 hole. This is illustrated in Fig. 12 by coolant pathlines from both rows of holes. The pathlines merge into a single stream, which becomes narrower with downstream distance because of the strong acceleration. Thus, as in the case of the REF configuration, there is a wide region directly aft of the Row1 hole that

**Fig. 5** Plots of predicted laterally averaged effectiveness for the CD, FF, and REF geometries of blowing ratios of (a) \( M = 1.0 \), (b) \( M = 1.5 \), (c) \( M = 2.0 \), and (d) \( M = 2.5 \)

---

**Fig. 6** Contours of \( V_y \) on the FHEP of the Row1 conical diffuser hole at \( M = 2.0 \) show a fairly low, uniform coolant flow

**Fig. 7** Lines of constant \( \theta \) on three planes of constant \( z \)-coordinate between Row1 hole centerlines at the blowing ratio \( M = 2.0 \). Note the extremely low trajectory of the coolant jet.
sees no coolant, and this fact was evident in the contours of effectiveness. Merging of the Row1 and Row2 coolant jets was present to some degree throughout the range of simulated blowing ratios.

### 4.2.1 Heating Within the Film Hole

From the contours of normalized $y$-velocity in Fig. 9, it was observed that crossflow blockage results in a large “dead” region at the upstream side of the Row2 FHEP where almost no coolant exits the hole. This is a prime condition for two phenomena that may cause thermal problems inside the film hole: crossflow ingestion and thermal diffusion. McGrath and Leylek [13] highlighted the potential risk of these mechanisms in simulations of film cooling from diffused holes at large compound angles. Crossflow ingestion arises from the action of a vortex that directs hot fluid below the FHEP in a region where coolant momentum is minimal. The typical ingestion vortex results when the vorticity in the crossflow boundary layer becomes oriented in the local direction of flow as flow wraps around the emerging jet (the “horseshoe” vortex). A clue that minor crossflow ingestion is present in the Row2 CDIFF film hole is the region of negative normal velocity near the LE side of the hole at $M \approx 2.0$ (Fig. 9). The horseshoe vortex can be identified in Fig. 13 by the coalescence and roll-up of the paths of zero-mass particles released in the crossflow boundary layer (0.1 mm above the surface) just upstream of Row2. The fluid is clearly observed.

![Fig. 8](image1.png) Contours of $V_y$ on a plane through the centerline of the Row2 film hole (from upstream to downstream edge) at $M = 2.0$ shows the development of a jetting region

![Fig. 9](image2.png) Lines of constant $V_y$ on the Row2 FHEP for (a) $M = 1.0$, and (b) $M = 2.0$ show the highly nonuniform velocity field at the diffuser exit

![Fig. 10](image3.png) Contours of $\theta$ on three planes of constant $x$-coordinate aft of the Row2 film hole for the blowing ratio $M = 2.0$. The dashed lines mark the extent of the region on the surface in which $\theta \leq 0.3$.

![Fig. 11](image4.png) Velocity vectors sized by in-plane velocity magnitude on the $x/D = 4.5$ plane (just aft of the Row2 hole) at $M = 2.0$ showing a relatively strong secondary flow. The gray line marks the $\theta = 0.5$ isotherm.

![Fig. 12](image5.png) Pathlines of the coolant from Row1 and Row2 for the case of $M = 2.0$ show the strong interaction between rows
Fig. 13 Paths of massless particles released in the crossflow boundary layer just upstream of Row2 for $M=2.0$ show the vortex that brings crossflow below the FHEP

to dip below the FHEP. Throughout the range of blowing ratios simulated, crossflow ingestion was very mild and the fluid that travels into the hole is at a moderately low temperature because it contains coolant from Row1. Also, the ingestion is located away from the film-hole wall, so it does not cause significant heating of the metal.

Diffusion of thermal energy from the crossflow into the coolant is a more important mechanism than secondary flow under the present conditions. This phenomenon occurs in areas where hot crossflow moves over stagnant coolant at the FHEP, and it is greatly aided by the high turbulence levels observed in the vicinity of the jet-mainstream interaction. In the case of $M=2.0$, most of the crossflow traveling over the hole is relatively cool from the Row1 coolant, and only near the TE the crossflow approaches $\theta = 1$. However, at $M=1.0$ most of the Row2 hole sees the maximum crossflow temperature. Additionally, at this blowing ratio the coolant is blocked over a larger portion of the FHEP, creating more surface area for diffusion to take place.

Figure 14 shows contours of temperature on the FHEP of Row2 for $M=1.0$ and $M=2.0$. At the low blowing ratio, the temperature increases toward the TE to a maximum of greater than $\theta = 0.8$, almost at the mainstream temperature. The heating is not as severe in the case of $M=2.0$, in which the normalized temperature exceeds 0.4 near the TE. The film-hole wall is located very close to the FHEP on this side of the hole, since the shallow injection angle and the diffusion combine to give a quite small film-hole wall angle at the TE. Thus, diffusion of thermal energy occurs right down to the wall inside of the hole. Contours of normalized temperature on the film-hole wall are shown in Fig. 15 for $M=1.0$ and 2.0. At $M=1.0$, wall heating is severe, with $\theta$ increasing rapidly to a value of about 0.9 at the TE. The effect is less severe at the higher blowing ratio. The heating of the metal may result in detrimental thermal stresses in this area of complex geometry and is likely an important design consideration.

5 Conclusions

A previously validated computational methodology was extended to evaluate the performance of exit-diffused holes in film cooling of a turbine airfoil leading edge. The conical diffuser holes gave a 10 to 40% increase in laterally averaged effectiveness as compared to the cylindrical holes over the complete leading edge, with the only exception being right at the location of the hole rows. This trend was observed through the range of simulated blowing ratios. The advantage in effectiveness was due to the shallower trajectory of the coolant exiting the holes, causing it to stay closer to the surface than in the case of nondiffused holes. However, the increased effectiveness comes at the expense of a tendency for heating of the Row2 film-hole wall near the TE, where the metal is very near the FHEP due to the small injection angle. The heating within the hole is due primarily to thermal diffusion in a low-momentum region that develops near the upwind side of the FHEP, and it could be problematic if not accounted for in the overall design. Finally, this work illustrates how a carefully designed computational program may be effectively used to conduct design iterations, in this case the analysis of an innovative film-hole geometry.

Acknowledgments

This research was supported by General Electric Aircraft Engines (GEAE) as part of the University Strategic Alliance (USA) program. The authors appreciate the assistance of D.C. Wisler, F.A. Buck, and M.L. Shelton at GEAE. The first author is grateful for funding from a National Science Foundation Graduate Research Fellowship. Appreciation also goes to Sun Microsystems for a grant of computer hardware and to Fluent, Inc. for software support.

Nomenclature

$\theta$ = wall angle at the TE
$\theta_0$ = free-stream temperature
$\Delta T$ = temperature difference
$\Delta T_{	ext{inlet}}$ = inlet temperature difference
$\Delta T_{	ext{wall}}$ = wall temperature difference
$\Delta T_{	ext{film}}$ = film temperature difference
$V_{	ext{in}}$ = inlet velocity
$V_{	ext{out}}$ = outlet velocity
$V_{	ext{total}}$ = total velocity
$V_{	ext{static}}$ = static velocity
$V_{	ext{kinetic}}$ = kinetic velocity
$V_{	ext{thermal}}$ = thermal velocity
$V_{	ext{molecular}}$ = molecular velocity
$V_{	ext{adiabatic}}$ = adiabatic velocity

Fig. 14 Contours of $\theta$ on the FHEP of the Row2 conical diffuser hole for the cases of (a) $M=1.0$, and (b) $M=2.0$

Fig. 15 Lines of constant $\theta$ on the film-hole walls inside the Row2 CDIFF hole at blowing ratios of (a) $M=1.0$, and (b) $M=2.0$ reveal the severe heating of the metal surface near the TE

$\text{AR} = \text{film-hole exit area ratio (CDIFF to REF)}$
$\text{CDIFF} = \text{referring to the conical diffuser film holes}$
$\text{D} = \text{film-cooling hole diameter [m]}$
$\text{DR} = \text{coolant-to-mainstream density ratio} = \rho_c/\rho_h$
$\text{FHEP} = \text{film-hole exit plane}$
$k$ = turbulent kinetic energy [m²/s²]
$L$ = film-cooling hole length [m]
$LE$ = film-hole leading edge
$M$ = global coolant to mainstream mass flux (blowing) ratio $= \rho_c U / \rho_u U_w$
$P$ = static pressure [N/m²]
$REF$ = referring to standard cylindrical film holes
$RKE$ = realizable $k$-$\epsilon$ turbulence model
$Row1$ = pertaining to row of film holes on stagnation line
$Row2$ = pertaining to downstream row of film holes
$S$ = film-cooling hole lateral spacing (within each row) [m]
$T$ = static temperature [K]
$TE$ = film-hole trailing edge surface
$U$ = local velocity [m/s]
$V_y$ = normalized $y$-velocity $= U_y / U$
$x$ = streamwise coordinate originating at stagnation line
$y$ = coordinate normal to leading edge surface
$y^*$ = non-dimensional distance from wall
$z$ = spanwise coordinate
$\alpha$ = injection angle with respect to leading-edge surface
$\phi$ = compound angle
$\epsilon$ = dissipation rate of turbulent kinetic energy [m²/s³]
$\eta$ = adiabatic effectiveness $= (T'_{aw} - T) / (T - T_m)$
$\bar{\eta}$ = laterally averaged adiabatic effectiveness
$\nu$ = kinematic viscosity [m²/s]
$\rho$ = density [kg/m³]
$\theta$ = dimensionless temperature $= (T - T_\infty) / (T_m - T_\infty)$

Subscripts
$\infty$ = mainstream conditions at crossflow inlet plane
$aw$ = adiabatic wall
$j$ = coolant jet (average) conditions
$y$ = $y$ direction component (normal to leading edge and FHEP)

References