ABSTRACT

Large scale computational analyses have been conducted and results compared with experiments to understand coolant jet and crossflow interaction in discrete-jet film cooling. Detailed three-dimensional elliptic Navier–Stokes solutions, with high order turbulence modeling, are presented for film cooling using a new model enabling simultaneous solution of fully coupled flow in plenum, film-hole, and cross-stream regions. Computations are carried out for the following range of film cooling parameters typically found in gas turbine airfoil applications: single row of jets with a film-hole length-to-diameter ratio of 1.75 and 3.5; blowing ratio from 0.5 up to 2; coolant-to-crossflow density ratio of 2; streamwise injection angle of 35 degrees; and pitch-to-diameter ratio of 3. Comparison of computational solutions with experimental data are in good agreement. Moreover, the current results complement experiments and support previous interpretations of measured data and flow visualization. The results also explain important aspects of film cooling, such as the development of complex flow within the film-hole in addition to the well known counterrotating vortex structure in the cross-stream.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>α</td>
<td>injection angle of coolant jet</td>
</tr>
<tr>
<td>D</td>
<td>diameter of coolant jet injection hole</td>
</tr>
<tr>
<td>DR</td>
<td>density ratio = ( \rho_j/\rho_\infty )</td>
</tr>
<tr>
<td>ε</td>
<td>dissipation rate of turbulent kinetic energy</td>
</tr>
<tr>
<td>η</td>
<td>adiabatic film effectiveness = ( (T_{aw}-T_\infty)/(T_j-T_\infty) )</td>
</tr>
<tr>
<td>I</td>
<td>momentum flux ratio = ( (\rho V_j^2)/(\rho V_\infty^2) ) = DR((VR))^2</td>
</tr>
<tr>
<td>k</td>
<td>turbulent kinetic energy</td>
</tr>
<tr>
<td>L</td>
<td>length of film-hole</td>
</tr>
<tr>
<td>L/D</td>
<td>length-to-diameter ratio of film-hole</td>
</tr>
<tr>
<td>M</td>
<td>blowing (or mass flux) ratio = ( (\rho V_j)/(\rho V_\infty) )</td>
</tr>
<tr>
<td>PD</td>
<td>pitch-to-diameter ratio of adjacent jets</td>
</tr>
<tr>
<td>T</td>
<td>local temperature</td>
</tr>
<tr>
<td>(T_{aw})</td>
<td>adiabatic wall temperature</td>
</tr>
<tr>
<td>(T_\infty)</td>
<td>mainstream temperature</td>
</tr>
<tr>
<td>(T_j)</td>
<td>coolant jet temperature</td>
</tr>
<tr>
<td>TL</td>
<td>turbulence intensity level</td>
</tr>
<tr>
<td>u,v,w</td>
<td>lateral, vertical, and streamwise components of velocity vector</td>
</tr>
<tr>
<td>(V_j)</td>
<td>magnitude of velocity vector</td>
</tr>
<tr>
<td>(VR)</td>
<td>velocity ratio = ( V_j/V_\infty )</td>
</tr>
<tr>
<td>(x,y,z)</td>
<td>lateral, vertical, and streamwise directions in Cartesian coordinates</td>
</tr>
<tr>
<td>(aw)</td>
<td>conditions at adiabatic wall</td>
</tr>
<tr>
<td>(\infty)</td>
<td>mainstream conditions at crossflow inlet plane</td>
</tr>
<tr>
<td>j</td>
<td>conditions at exit plane of film-hole</td>
</tr>
<tr>
<td>p</td>
<td>conditions in coolant supply plenum</td>
</tr>
</tbody>
</table>

1.0 INTRODUCTION

Modern aircraft gas turbine hot section components operate in extremely harsh environments where ambient temperatures routinely exceed the melting point temperature of available alloys. Reliable operation and prolonged useful life of components require controlling both temperature level and gradient. Discrete-jet film cooling, often used in conjunction with internal convective cooling, is a popular method available to designers of such components. Although there are many configurations of film cooling, one common feature is the highly complex nature of the flowfield created by a coolant jet interacting with a hot cross-stream.

Current flowpath heat transfer design practice is mostly empirical in nature and relies heavily on a large experimental data base.
However, it is inconceivable to have a large enough data base for all possible combinations of geometric and flow parameters. Yet, recent design trends toward even higher cycle pressure and temperature with drastically reduced coolant supply have rapidly moved present day designs away from existing data bases. In addition, design cycle time is being compressed in order to beat the competition to the market. Thus, there is an urgent need to reduce the level of empiricism in heat transfer design practice and to develop a truly predictive capability for film cooling.

There have been significant advances in all facets of computational fluid dynamics methods for complex fluid flow problems in the last two decades. Concurrently, storage capacity and computation speed of computers have increased by many orders of magnitude with accompanying decreases in the cost of computation. Proliferation of supercomputers has made these machines readily available for many. In light of these advances, a real predictive capability for film cooling heat transfer is now feasible.

2.0 LITERATURE REVIEW

For the purpose of the following discussion, the large body of film cooling papers in the open literature is divided into the broad categories of experiment and computation. Experimental works, in turn, are split into two groups. The first group includes studies which generate “surface” measurement data for use in design activities. These data are presented most commonly in the form of adiabatic film cooling effectiveness (η). Other studies that support computational model development activities and provide detailed “field” measurements are discussed in the second group. Computational works are also split into two groups. Two-dimensional boundary layer models, relatively simple and fast-running codes which provide circumferentially averaged results when calibrated against a suitable database, are presented in one group. The final group includes three-dimensional Navier–Stokes models which generate detailed results and have the greatest potential for true predictive capability.

2.1 Experimental Studies

2.1.1 Surface Measurements

There are an abundance of surface measurement data in the open literature for design use especially for slot, transpiration, and single hole injection configurations. Most of these and the early single row of discrete–jet experiments were conducted with coolant jet–to–crossflow density ratio (DR) close to unity. High DR cases are more relevant for gas turbine applications and such studies have been carried out at many institutions. A short list of representative investigations in this category include the works of: (i) Petersen, Eckert, and Goldstein (1977), (ii) Foster and Lampard (1980), and (iii) Forth and Jones (1988). Although these and other studies have covered a wide range of density and blowing ratios, one feature they all had in common was their use of large film–hole length–to–diameter ratio (L/D) ranging from about 10 to 40, configurations atypical of gas turbine airfoils.

Sinha, Bogard, and Crawford (1990) reported adiabatic effectiveness for a large number of density ratio (DR=1.2, 1.6, and 2) and blowing ratio (M=0.25 to 1.0) combinations for a single row of 35 degree holes with PD of 3. A unique feature of this study is its use of short film holes resulting in a small film–hole length–to–diameter ratio of L/D=1.75. The configuration used in these experiments was replicated exactly in one of the two computational models reported in the present study.

2.1.2 Field Measurements

The number of studies available in the open literature on field measurements are far less than those for surface data. The instrumentation used to survey the field is critically important in these types of measurements and they were found to range from pitot probes in early studies, to single, X–sensor, and triple–sensor hot–wire anemometry, and finally to three–beam laser–doppler velocimetry (LDV) used in the latest publications. A brief review of the most significant papers for our purposes is provided next.

Crabb, Durao, and Whitelaw (1981) studied the hydrodynamics of a normal jet interacting with a crossflow using a X–sensor hot–wire probe in the far field and LDV in the near field. In these tests the density ratio was kept at unity with a blowing ratio of 1.5 and 2.3. Results indicate a highly disturbed velocity profile at the jet exit plane. The jet supply pipe was long, as commonly found in almost all early studies, with L/D=30.

The present authors benefited greatly from a series of papers published by a group headed by Prof. Rodi at the University of Karlsruhe in Germany. Andreopoulos and Rodi (1984) documented what was then the most detailed study of the turbulence field for a normal jet in a crossflow. Density ratio was kept at unity and blowing ratio at 0.5. The measurements clearly showed the presence of a massive blockage occupying the upstream portion of the jet exit area. The jet was forced to leave mostly from the downstream half of the exit plane. Consequently, the velocity profile taken at the centerline of the jet exit plane was highly skewed with its maximum point shifted downstream from the center. Disturbances created by the interaction of jet and crossflow were found to penetrate into the supply pipe and influence the flow upstream of the exit plane. This configuration, too, had a long supply pipe with L/D=12.

A row of inclined jets at a number of injection angle, density ratio, and blowing ratio parameter combinations were studied by Kadotani and Goldstein (1979), Yoshida and Goldstein (1984), Le Brocq, Launder, and Priddin (1973), Launder and York (1974), Foster and Lampard (1980), and others. Again, a common feature of all these studies was their use of long coolant supply tubes with L/D ranging from about 11 to 62.

Studies conducted by the University of Texas group represent to our knowledge the most comprehensive survey of inclined jets–in–crossflow. Pietrzyk, Bogard, and Crawford (1989a) documented jets with density ratio of unity and in a companion paper the same authors (1990) published results for density ratio of 2. Both studies were done on the same film configuration with a single row of jets inclined at 35 degrees and spaced 3 diameters in the lateral direction. The blowing ratio was 0.25, 0.5, and 1 in the first paper and 0.5 in the second. Aside from a cryogenically cooled jet, this particular facility is set apart from all others because of its ability to accommodate models with a small film–hole length–to–diameter ratio of 3.5. The low L/D makes this study very relevant to gas turbine airfoil applications. Detailed measurements were reported of the distributions of streamwise and normal components of the mean and fluctuating velocity.
along with the dominant Reynolds stress from one diameter upstream to 30 diameters downstream of the hole. A three-beam laser doppler velocimetry system enabled almost simultaneous measurements of all three velocity components for accurate turbulence data. Two observations were made which are most interesting and unique in the open literature. First, as the blowing ratio varied from 0.25 to 1.0, the location of the maximum in the centerline velocity profile at jet exit shifted from downstream to upstream of the center. Second, at increased blowing ratio a high level of turbulence was measured in the downstream half of the jet exit plane. The authors explained these two observations by hypothesizing the presence of a possible separation region within the film hole. One of the two 3-D Navier-Stokes models presented in the present study is a detailed replica of the configuration described in these papers.

2.2 Computational Studies

2.2.1 2-D Boundary Layer Models

A number of 2-D boundary layer type computer programs that predict laterally averaged behavior of film cooling are described in the literature by Crawford, Kays, and Moffat (1980), Tafti and Yavuzkurt (1990), Norton et al. (1990), and Haas, Rodi, and Schonung (1991). These codes are intended for parametric runs in preliminary design type settings. By nature they are relatively simple, fast-running, and very economical. A suitable data base, either experimental or computational, is needed in order to calibrate these models. It is then hoped that the calibrated model will not only reproduce the entire data base faithfully but, in fact, successfully interpolate to fill in the gaps in the data base and perhaps even extrapolate beyond it. Clearly, 2-D boundary layer models cannot predict film cooling in a totally new configuration or an existing geometry placed in a substantially different flow regime.

2.2.2 3-D Navier-Stokes Models

There are a limited number of entries in the open literature on 3-D Navier-Stokes models to predict film cooling. As stated above, these models have the greatest potential to reduce the level of empiricism and provide a true predictive capability in design tools. However, along with the increased level of sophistication comes an order of magnitude increase in the required computational resources.

The original "fully-parabolic" scheme applied to jet-boundary-layer interaction region by Tatchell (1975) resulted in physically unrealistic solutions, thus providing an early indication of strong deviation from boundary-layer-like flows.

Bergeles, Gosman, and Launder (1978) documented a "semi-elliptic" (or "partially-parabolic") scheme used in conjunction with an anisotropic turbulence model for an isolated jet injected into a crossflow at 90 and 30 degrees. Their results indicated reasonably good agreement with measurements as long as blowing ratio remained extremely low. For example, for the case with a 90 degree injection angle, it proved to be impossible to obtain fully converged solution at blowing ratio of 0.2. Similarly, the results deteriorated rapidly beyond M=0.5 in 30 degree jet simulations. These authors discovered that the relative insensitivity of the results to the details of the injectant profiles at the exit plane did not apply at higher M when the jet was on the verge of lift-off. The predicted flow character just downstream of the jet was poor even after altering the jet exit plane treatment, thus leading to a conclusion that "evidently there were more important factors involved." In closing, the authors indicated that this situation could perhaps be remedied by extending the numerical computations some distance into the injection hole itself.

A total of 27 test cases were computed for a single row of holes by Demuren, Rodi, and Schonung (1985) using the "locally-elliptic" procedure of Rodi and Srivasta (1980). In this model, the fully-elliptic treatment was applied only to a small region containing flow reversal; however, in the near and far fields the model reverted to partially-parabolic and fully-parabolic schemes, respectively. Also included in the model was the extension of Bergeles et al. (1978) to account for anisotropic eddy viscosity and diffusivity. The authors acknowledged the significant influence of the crossflow on the jet exit profiles. However, indicating a lack of sufficient experimental information they assumed uniform velocity at the jet exit plane.

White (1980) presented a fully elliptic treatment for an isolated normal jet injected into a crossflow. The novel feature of this model was its use of separately computed flowfields in the pipe and cross-stream regions which were linked via constant total pressure assumption across the jet exit plane. Comparison of jet exit profiles with measurements showed good agreement; however, the method was found in that study to be restricted to normal jets supplied from tubes with large length-to-diameter ratios. An encouraging observation from that study and another one by Patankar, Basu, and Alpay (1977) for a dilution jet involves the ability of a fully elliptic scheme to handle strong jets that are far beyond lift-off.

2.2.3 Remarks

The 3-D models cited above suffer from a number of serious deficiencies. For example, the flow simulations are confined to the crossflow region only. This region is represented using a simple Cartesian grid where the jet exit geometry, typically round or elliptic depending upon jet injection angle, is approximated as an "equivalent area" in a stair-step manner. More critically, various assumptions used to obtain boundary conditions at the jet exit plane in the form of exit profiles are, at best, poor approximations for those cases with large length-to-diameter ratios. They would be even worse when L/D is small. According to measurements made in recent experiments by Pietrzyk (1989), the use of such profiles in configurations more representative of gas turbine airfoil applications cannot be justified.

3.0 THE PRESENT CONTRIBUTION

There were two primary objectives of this work. The first was to develop a 3-D Navier-Stokes model to predict film cooling in configurations relevant to gas turbine airfoils. The second was to compare the results from these predictions to experiments in order to explain measured data and flow visualization observations. The notable features of the present study, which have not been considered in any previous investigation, are:

a. Fully-coupled and elliptic computation of flow in plenum, film-hole, and cross-stream regions of a film cooling situation.
b. Detailed description of the complex nature of flow within a small length-to-diameter ratio film-hole and mechanisms responsible for unusual profiles at the jet exit plane.

c. Exact representation of an inclined, round film-hole geometry using a highly orthogonalized grid mesh obtained from a state-of-the-art solid modeler integrated with a generalized, non-orthogonal curvilinear coordinate system grid generator.

d. Description of a computational strategy to obtain fully converged results for blowing ratios as high as 2.0 in a multi-zone film cooling domain.

4.0 TECHNICAL APPROACH

The overall approach used to achieve the above objectives is as follows: (i) The computational effort made full use of data obtained in a test program setup at the University of Texas at Austin by Profs. Bogard and Crawford and their students to conduct fundamental studies of jets-in-crossflow in large-scale film cooling configurations relevant to gas turbine airfoils. (ii) A three-dimensional, elliptic Navier–Stokes model was developed to compute the flowfield in a single row film cooling situation in a fully-coupled manner. Two relatively fine computational grid meshes were generated on two different film cooling geometries which resulted in large-scale simulations of film cooling flowfields for a single density and a number of blowing ratios. (iii) The results were used to understand important aspects of flow in film cooling.

5.0 3-D NAVIER–STOKES SYSTEM

The Navier–Stokes system used to obtain the computational results presented in this paper is described briefly. This system was created by modifying, extending and then integrating together a number of readily available, suitable codes into a computational film cooling prediction tool for use in gas turbine applications. It is comprised of the following major modules: geometry generator, grid generator, pre-processor, CRAY JCL generator, solver, and post-processor. The geometry and grid generator modules represent a breakthrough in our ability to capture a complex film cooling configuration and generate a high quality computational grid. This capability is critically important because the quality of a computational solution is strongly linked to the quality of the grid mesh.

5.1 Computational Elements

A state-of-the-art solid modeler, I-DEAS, available commercially from Structural Dynamics Research Corporation (SDRC), was integrated into the computational system as a geometry generator. This approach was selected because it appears to be most promising especially in film cooling applications employing shaped-holes on gas turbine airfoils.

The grid generator creates a grid mesh within each sub-volume of the entire computation domain passed to it by the geometry modeler. All sub-volumes are then combined and grid nodes re-numbered so as to form a single consistent cluster for a structured grid solver.

The pre-processor, solver, and post-processor modules employed in the present system were essentially the PHOENICS system of codes described by Ludwig, Qin and Spalding (1989) with suitable modifications and a JCL generator to run it on a CRAY Y/MP supercomputer. The solver is based on a finite-volume discretization scheme and uses a staggered grid as described by Patankar (1980).

5.1.1 Turbulence Model

Turbulence closure is attained by the use of the standard k-ε model employing the generalized wall function treatment described by Launder and Spalding (1974). It is important to note that absolutely no special handling, tuning, or adjustment of any kind was made to the turbulence model, or any other part of the code, to “match” experimental data in the present study. Our intent is to determine how well the standard k-ε model can perform in an elliptic, fully-coupled simulation of film cooling jets and to remedy its shortcomings in future studies.

5.2 Validation of Computational System

The 3-D Navier–Stokes system has undergone extensive inhouse testing by applying it to relatively simple 2-D to more demanding 3-D problems for which code validation quality data exist. The battery of validation test cases includes the following: (i) heat transfer on smooth surfaces, (ii) flow over a backward-facing step, (iii) two-dimensional normal slot-jet injection, and (iv) 3-D channel flow heat transfer. Overall, the results obtained in these test cases were found to be satisfactory.

6.0 EXPERIMENTAL TEST FACILITY

A complete description of the experimental facility, including the test section, cryogenic cooling system, and instrumentation used, is given by Pietrzyk (1989). The facility was used to make detailed measurements of mean and turbulence characteristics of jets-in-crossflow along with adiabatic film cooling effectiveness for injection from a single row of 35 degree holes, laterally spaced 3 diameters apart, on a flat test surface as shown in Figure 4.1. The row of holes are placed 19 diameters downstream of the leading edge of the test plate and data are obtained from 1 diameter upstream to 30 diameters downstream of the holes.

Two separate series of tests were run. In one of the series, Pietrzyk, Bogard, and Crawford (1989a, 1990) documented the hydrodynamics of jet/crossflow interaction using a test plate with a hole length-to-diameter ratio of 3.5. In these tests density ratio was set at either 1 or 2, and blowing ratio varied from 0.25 to 1. In the other series, Sinha, Bogard, and Crawford (1990) acquired adiabatic effectiveness data on a test plate with L/D of 1.75 where density and blowing ratios were varied from 1.2 to 2 and from 0.25 to 1, respectively.

The boundary layer growing on the bottom surface of the wind tunnel was sucked out and a uniform starting velocity profile created at the leading edge of the test plate. Measured data indicate that a turbulent boundary layer commenced immediately downstream of the test plate leading edge, due perhaps to the presence of a small separation bubble there, Crawford (1992). This is believed to be the main reason for the excellent agreement between the predicted and measured boundary layer thickness at 1 diameter upstream of the film holes.
7.0 DETAILS OF COMPUTATIONAL MODEL

Details of the computational model, corresponding to its experimental counterpart described above, are presented here. All dimensions are measured in a Cartesian coordinate system with its origin placed at the upstream edge of the film hole on the top surface of the test plate and its x, y, and z axes aligned with the lateral, vertical, and streamwise directions as seen in Figure 1.

7.1 Geometry

Two exact replica solid models of the test facility including the plenum, film-hole, and cross-stream regions were constructed using I-DEAS. The only difference between the two models was the film-hole length-to-diameter ratio L/D. The first model had an L/D of 3.5 and the second 1.75, just as in the experiments. Details of the plenum construction not found in Pietrzyk (1989) were obtained from Crawford (1991) and incorporated into these models. Also, edges of the film hole at the top and bottom surfaces of the test plate were made sharp as indicated in the description of the experimental test facility. As shown in Figure 1, overall extent of the computation domain in the lateral, vertical, and streamwise directions was 1.5D, 16D, and 49D, respectively.

![Figure 1. Essential features of experimental film cooling configuration showing overall extent of computation domain and coordinate system.](http://proceedings.asmedigitalcollection.asme.org/doi/abs/10.2514/6.1993-3538)

7.2 Grid Mesh

A highly orthogonalized, non-uniform, fine grid mesh was generated with grid nodes clustered in the immediate vicinity of the discrete film cooling jet. A total of 200,090 grid nodes were used with 22 nodes in the lateral, 85 nodes in the vertical, and 107 nodes in the streamwise directions. The most difficult to grid and yet the most critical portion of the entire mesh is the region within the film hole itself. A highly enlarged view of the grid mesh in this region on the hole centerline plane is shown in Figure 2a. The film hole was divided into four sub-volumes stacked in the vertical direction in such a way that the center two volumes naturally contained an orthogonal grid as seen in this figure. Grid nodes in the other two sub-volumes at the inlet and exit regions of the film hole were individually orthogonalized with the use of surface and volume optimizers. Figure 2b shows the top view of the orthogonalized grid at the exit plane of the round film-hole with a semi-elliptic break-out. The semi-elliptic cross-section was transformed into a purely orthogonal rectangle within a short distance above and below the test plate.

![Figure 2. Computational grid for 3-D discrete jet film cooling: 200,090 total grid nodes with 22 lateral, 85 vertical, and 107 streamwise nodes.](http://proceedings.asmedigitalcollection.asme.org/doi/abs/10.2514/6.1993-3538)

Other important grid parameters were conservatively set and tightly controlled. For example, the stretching ratio used in all three directions in the whole domain was kept well within 20 percent. Similarly, the grid aspect ratio, especially near the coolant jet, was kept well under 10. Finally, proximity of nodes near metal surfaces was carefully adjusted so that average y+ was about 30 inside the plenum, ranged from 30 to 100 within the film hole, and 50 in the crossflow as M varied from 0.5 to 2.

7.3 Boundary Conditions

Boundary conditions are prescribed at all six boundary surfaces of the computation domain by imposing exactly the measurements made in experiments. Mainstream conditions were kept the same in all cases and the coolant flow rate was altered to change the blowing ratio in a way fully consistent with the procedure described by Pietrzyk (1989). In the lateral (x) direction, both bounding surfaces are symmetry planes; therefore, the x-direction component (u) of the velocity vector and normal gradients of all other dependent variables are prescribed as zero.

In the vertical (y) direction, at the bottom surface, which sits on the flow straighteners, coolant mass flux, enthalpy, and the y-direction component (v) of the velocity are fixed to produce the...
desired blowing ratio at the known plenum temperature ($T_p$) of 153 K. The remaining two components of velocity were set to zero. Similarly, assuming a condition of local equilibrium, a uniform distribution of $k$ and $\varepsilon$ were computed from velocity, turbulence intensity of 0.2 percent, and length scale equal to $1/10$ of plenum width. All parameters affected by pressure were adjusted during the computation cycle as the correct plenum pressure emerged while iterations continued and the solution proceeded toward convergence. At the top surface, measured data indicate that a "slip-wall" type boundary condition should work properly due to sufficiently far placement of this surface from the test plate. Therefore, a slip condition was imposed by setting the $v$-component of velocity and normal gradients of all other dependent variables to zero.

In the streamwise ($z$) direction, at the inlet plane, cross-stream mass flux, uniform enthalpy, and $z$-direction component ($w$) of 20 m/s were specified to produce the desired mainstream Reynolds number at the measured crossflow temperature ($T_c$) of 302 K. The remaining two components of the inlet velocity vector were set to zero. It was possible to prescribe a "plug-flow" velocity profile because the inlet plane was placed at the leading edge of the test plate where a new boundary layer was triggered. Also, a uniform inlet $k$ and $\varepsilon$ profiles were imposed following a procedure similar to that described for the boundary conditions at the plenum inlet plane except that turbulence length scale was $1/10$ of computational passage height. At the exit plane, pressure level was specified along with zero streamwise gradient for all other dependent variables.

### 7.4 Initialization Strategy

Initialization of all the dependent variables in the entire field turned out to be a critical item, especially at high $M$, in securing converged solutions in the fully-elliptic and fully-coupled computation of the complex film cooling situations presented in this paper. First, the three regions, namely, plenum, film-hole, and cross-stream, were separately initialized as accurately as possible for the lowest blowing ratio ($M=0.5$) case. Next, the fully converged solution obtained from this run was used as the initial field for the next case with a slightly higher $M$, and so on. This strategy made it possible to successfully converge the computational solution for the highest blowing ratio ($M=2$) attempted in this study.

### 7.5 Convergence

Starting from a suitable initial distribution for all field variables, iterations were continued until each transport equation satisfied a built-in convergence criterion. The computational solution was declared "fully converged" when the normalized residual of each governing equation was at or below 1 percent level. The normalized residual was computed by adding the absolute values of the residuals for all the finite volumes in the whole domain and dividing this sum by the total inlet flux (crossflow and plenum combined) of a relevant quantity for each equation (e.g. enthalpy flux for the energy equation).

### 8.0 COMPUTED 3-D FLOWFIELD

Simulation details and comparisons of computed flowfield with experiment are presented in this section. Unless otherwise noted, all results presented here, except for those for adiabatic effectiveness, are for the case with $L/D=3.5$.

#### 8.1 Details of Flow in Film Hole

The complex nature of flow within the film-hole itself is described in this sub-section. This, to our knowledge, is the first time that any information, either experimental or computational, regarding the nature of flow in a film-hole with a small $L/D$ is documented in the film cooling literature.

Computed velocity vectors on the centerline plane of the film-hole are presented in Figures 3(a,b) for $M=0.5$ and $M=2.0$ cases, respectively. Clearly, no part of this flow resembles a fully-developed pipe flow as would be the case in a long coolant supply tube. Instead, a low momentum region appears near the inlet and downstream wall of the film-hole at $M=0.5$, Figure 3a. The coolant flow appears to be showing a slight jetting effect near the upstream wall. At the highest blowing ratio of $M=2$, shown in Figure 3b, a strong jetting effect is seen near the upstream wall.
with the coolant hugging this surface as it rides over the low momentum region near the downstream wall.

A view of the velocity vectors on a plane perpendicular to the general direction of the through-flow in the film-hole reveals the exact nature of the coolant flow in this region, Figures 4(a,b). The exact location of this plane within the film-hole is indicated by the two bold lines in Figure 2a. In this view, the crossflow direction above the jet exit plane is from left to right. Two counter-rotating vortices are present in the film-hole at both blowing ratios with their cores close to the downstream wall of the coolant supply tube. The vortex strength is directly related to blowing ratio as it is much higher in the high M situation shown in Figure 4b than the low M case seen in Figure 4a. The vortex structure seen here is due to the secondary flow created by a large change in the direction of the coolant flow as it enters the inclined supply tube from the plenum. The same mechanism that induces secondary flow in curved tubes is responsible for the creation of vortices within the film-hole. The sense of rotation of the vortices is the same as a similar counter-rotating vortex structure in the cross-stream to be discussed shortly. Flow visualization experiments conducted at the University of Texas confirm the existence of the counter-rotating vortex structure shown here, Crawford (1992).

The final observation, which is important for our discussion in Section 8.3.1 below, involves the production of turbulence inside the film-hole. Computational results indicate that as the blowing ratio increases, the coolant supply tube becomes the major source of turbulence. The centerline turbulent kinetic energy contours for M=2 presented in Figure 5 reveal that the location of the peak level is indeed within the film-hole. Previous results, presented in Figures 3 and 4, are helpful in explaining this observation. As M increases, the counter-rotating vortex strength increases and is accompanied by strong jetting of coolant over the vortices. The highest velocity gradients are found in the interface region between the jet and the vortex structure. It is precisely at this interface that high turbulence is generated.

8.2 Details of Flow at Exit Plane of Coolant Jet

In this sub-section we focus our attention on the distribution of dependent variables at the coolant jet exit plane because other numerical studies published in the open literature have concluded that downstream results are very sensitive to the profile assumptions made there.

High L/D – The previously accepted view of exit profiles of coolant jets was based on either film cooling or jets-in-crossflow experiments conducted with unrealistically long supply tubes. Here, a single mechanism, namely the relative magnitudes of crossflow and coolant jet momentum, is thought to be the final arbiter of the profiles at the exit plane. According to this view, at low blowing ratios, say M=0.25, the coolant jet exits mostly through the downstream half of the exit plane due to blockage caused by the the crossflow carrying much higher momentum. The result is a skewed velocity profile with its peak shifted
downstream of the center as depicted in Figure 6a. At much higher blowing ratios, say \( M=4 \), the coolant jet exits as though the crossflow does not exist. This time, the exit velocity profile is very much like a fully developed turbulent pipe flow profile shown in Figure 6a. Of course, no variation is prescribed in the lateral direction.

\[ v_j = \frac{M}{2} \]

\[ M = \frac{2}{5} \]

\[ M = 0.5 \]

\[ 0 \leq y/D \leq 1 \]

\[ 0 \leq z/D \leq 1 \]

\( v_j \) is the jet velocity, and \( M \) is the blowing ratio.

Figure 6. Exit plane velocity profiles of film cooling jets at various blowing ratios.

This view, and hence the profiles resulting from it, when applied in a typical gas turbine film cooling configuration with a small L/D, may be highly misleading and is perhaps one of the prime causes of anomalous results, Jubran (1989), and Amer, Jubran, and Hamdan (1992).

Low L/D – The current computational results point to three mechanisms competing to control the profiles of coolant jets at exit plane. These are: (i) relative magnitudes of crossflow and coolant jet momentum, (ii) strength of jetting effect near upstream wall of film-hole, and (iii) strength of counter-rotating vortex structure. These mechanisms explain experimental measurements of Pietrzyk et al (1989a, 1990) and the current computational results at the exit plane are fully consistent with their data.

Velocity profiles measured by Pietrzyk (1989) along the centerline at the exit plane at \( M=0.25 \) show some crossflow blockage induced skewness. However, unlike previous measurements, the blockage is not excessive because it is offset by a relatively weak jetting effect present even at such low \( M \), Figure 3a. Experiments indicate a fully developed profile at a blowing ratio of only 0.5. This means that the crossflow blockage is fully offset by the presence of coolant jet at the upstream wall of the film-hole. At higher blowing ratio, such as presented in Figure 3b for \( M=2 \), the jetting effect overpowers the crossflow blockage and results in a velocity profile skewed upstream, Figure 6b. The role of the experimentally confirmed vortex structure is to create lateral variation and impart swirl to the coolant jet.

The same authors report data showing high turbulence levels spewing out of the downstream half of the exit plane at high blowing ratios. They hypothesized the existence of a separation bubble and a reattachment region within the film hole to explain their data, Pietrzyk et al. (1989a). This too is explained by the present simulations as turbulent kinetic energy contours shown in Figure 5 indicate the true source of turbulence (see discussion at the end of Section 8.1).

8.3 Details of Flowfield within Cross-stream

There are many consequences when a coolant jet interacts with a hot crossflow. In this sub-section, it is our aim to document the details of this interaction and the resulting flowfield in the cross-stream using data from computations and experiments.

8.3.1 Crossflow Results On Centerline Plane

Computed velocity vectors on the hole centerline plane near the jet exit are presented in Figure 7. These profiles are remarkably similar to those measured by Pietrzyk et al. (1989a) and have two important features that are noteworthy. First, the profile at \( z/D=5 \) indicates that the greatest streamwise momentum deficit is away from the test surface at about \( y/D \) of 0.5. Second, right above the momentum deficit zone there is a small region with a slight velocity overshoot created by the crossflow accelerating over the coolant jet.

One of the important indicators of the effectiveness of a film cooling jet is its trajectory. A low trajectory results in a high film effectiveness when a coolant jet is attached to the surface to be protected. A high trajectory, on the other hand, implies penetration of the jet into the crossflow and away from the injection surface, therefore, low film effectiveness. A scalar field, such as temperature, can be used as a tracer to determine the trajectory of a coolant jet.

Figures 8(a,b) show temperature contours for a jet with \( DR=2 \) and \( M=0.5 \) and 2, respectively, to demonstrate the effect of blowing ratio on jet trajectory. The coolant hugs the test surface at the low blowing ratio case presented in Figure 8a. A strong jet penetration, therefore, a high trajectory results as the blowing ratio increases to \( M=2 \), as seen in Figure 8b.

Experiments also indicate that blowing ratio influences the source of turbulence in film cooling situations such as the one considered here, Pietrzyk (1989). At low M, centerline turbulent kinetic energy contours in Figure 9a indicate that the shear layer between the crossflow and coolant jet is the main source of turbulence. This can be explained by noting that the difference...
in the streamwise momentum content of the jet and crossflow is highest at low M. At high M situations, Figure 9b, especially at small injection angles, this difference diminishes and the film–hole becomes the main source of turbulence for the reasons presented in Section 8.1.

8.3.2 Crossflow Results On Planes Normal To Cross–stream

More details of the complex nature of flow structure in the cross–stream are revealed by results plotted on planes perpendicular to crossflow. All views in the series of plots presented in this section are aft–looking–forward.

Two of the most important features in the cross–stream, namely, the counter–rotating vortex structure and kidney–shaped cross–section of coolant, are captured in Figures 10(a,b) by superimposing velocity vectors on temperature contours on a plane located 5 diameters downstream of film–hole leading edge. Apparently, the strength of secondary flow is again directly related to blowing ratio as was the case within the film–hole. This is evidenced by the low magnitudes of secondary flow velocity vectors in Figure 10a for M=0.5, and high ones in Figure 10b for M=2.0. The vertical and lateral location of the core center in Figure 10a agrees well with experimental data presented in Figure 10c for the same case. The net aerodynamic effect of the vortices interacting in such close proximity is to bring the individual cores laterally closer together and to lift the entire coolant film vertically away from the surface at high M. Another action of the vortex motion is the very undesirable outcome that hot crossflow is forced down and under the film layer. A kidney–shaped cross–section of the coolant jet is the final consequence of all the interactions reported here.

An isometric view of temperature contours on many crossflow planes is presented in Figures 11(a,b) for M=0.5 and M=2.0. These figures show clearly the differences in the trajectories of these two jets as they diffuse laterally and convect in the streamwise direction. It appears that computed jets do not seem to be diffusing as much as they should. This assessment is based on the observation that the limiting contours shown on the last plane at z/D=25 do not extend to the symmetry plane. However, contour plots of various parameters presented by Pietrzyk (1989) indicate merging of adjacent jets further upstream of this location. More on this later.

A similar isometric view of computed turbulence level contours for M=0.5 and M=2.0 is presented in Figures 12(a,b) along with experimentally measured data for M=0.5 shown in Figure 12c. There is a striking similarity between the shapes of computed and measured contours for M=0.5 shown in Figures 12a and 12c. Furthermore, quantitative agreement is also quite good. As discussed in previous sections, the shear layer between crossflow and coolant jet is the main source of turbulence at M=0.5, but the film–hole becomes the dominant source of this quantity at M=2.

8.4 Adiabatic Film Cooling Effectiveness

In this section we document the streamwise and lateral variation of film cooling effectiveness for the case with L/D=1.75 because experimental data are available on this configuration only.

Figure 8. Computed centerline temperature contours demonstrating effect of blowing ratio on trajectory of coolant jet.

Figure 9. Computed centerline turbulence intensity contours showing change in turbulence source with blowing ratio.
also that all streamwise distances are measured from the trailing edge (z/D=0) of the film-hole at the exit plane.

Computed and measured distribution of $\eta$ along the jet centerline for three blowing ratios (M=0.5, 0.8, and 1.0) are presented in Figures 13(a–c). Although the overall streamwise trend is correct, the predicted values are consistently high at $M=0.5$, Figure 13a. This situation is improved downstream of z/D=5 for $M=0.8$ and much improved for $M=1.0$ predictions shown in Figure 13b and Figure 13c, respectively. It appears that the computed film jet stays attached even at the highest blowing ratio shown as $\eta$ is always 1 near z/D=0. It is for this reason that bumps in the measured $\eta$ curves related to jet detachment-reattachment are missed by predictions.

Comparison of computed lateral variation of $\eta$ with that of experiments at five streamwise stations, shown in Figure 14, reveals one serious consequence of the insufficient rate of spreading noted in Section 8.3.2 above. Measured data indicates non-zero $\eta$ at the mid-pitch location (x/D=1.5) for all lateral profiles.
Figure 12. Isometric view of computed and measured turbulence intensity contours on crossflow planes shows remarkable similarities.

Figure 13. Streamwise variation of adiabatic effectiveness from computations and experiments along jet centerline.
except the one for z/D=1. Computed η curves, on the other hand, show almost zero effectiveness beyond about x/D=1.2 at all streamwise stations. Also, note that predicted centerline (x/D=0) η is higher at every station than measurements. This is consistent with our earlier observations on centerline effectiveness data for M=0.5.

9.0 DISCUSSION

This discussion focuses on the strongly coupled and fully-elliptic nature of film cooling flows in gas turbine configurations with small L/D, the complexity of flow within both the film-hole and the cross-stream, and the anisotropic behavior of turbulence created by jets-in-crossflow.

Flow in the film cooling configurations considered in this paper is strongly coupled and fully-elliptic. Previous studies have concluded that film cooling results downstream of the coolant injection location are very sensitive to the distribution of dependent variables at the jet exit plane. However, due to the close proximity of the inlet and exit planes, these exit plane conditions, in turn, are dependent upon conditions at the inlet plane of the film-hole. Therefore, the coupling is, by nature, three-way as it involves interaction of flow between the plenum, film-hole, and cross-stream regions. The strength of coupling increases as injection angle and L/D decrease and M increases. Also, flow both within and outside the film cooling hole is fully elliptic as seen in our results. This observation establishes the need for fully-elliptic treatment of such flows and signals that it is about time to abandon both fully– and partially-parabolic approaches.

There is a surprisingly complex flow within the film-hole itself. A counter–rotating vortex structure is generated in the hole which imparts swirl to the coolant jet as it is discharged into the hot crossflow. This vortex structure, together with the jetting effect created as the coolant is forced to accelerate over it, generates complex conditions at the jet exit plane. In fact, some of the measured and computed profiles have exactly the opposite trends of those obtained in experiments with large L/D.

Jets–in crossflow create one of the prime examples of an anisotropic turbulence field. We are fully aware of the inadequacies of eddy–viscosity type turbulence closure models to deal with these types of flows. We believe, however, that even if a perfect turbulence model existed, one could not obtain film cooling solutions with high accuracy the way that such flows have historically been modelled in the open literature. The improved 3–D Navier–Stokes model presented in this paper eliminates the need to specify boundary conditions at the jet exit plane, thus removing one of the main obstacles in the successful simulation of jet–crossflow interactions in film cooling. Our results show that using a computational model with a fully elliptic solution procedure to compute, simultaneously, the strongly coupled flow in the plenum, film-hole, and cross-stream regions can enable one to capture and explain much of the flow physics in discrete–jet film cooling at high blowing ratios. Thus, the stage is set for more accurate film cooling predictions with the use of a turbulence model which is capable of dealing with an anisotropic field. The lack of lateral spreading of the coolant in our results is caused by the inability of the k–ε model to cope with non-uniform rates of diffusion in different directions. It is to this task that we plan to turn next.

10.0 CONCLUSIONS

The following key conclusions about discrete–jet film cooling with a film-hole length–to–diameter ratio typical of gas turbine airfoils are drawn from three–dimensional Navier–Stokes analyses and experiments.

- The entire flowfield was dominated by a strong three-way coupling caused by interaction between the plenum, film-hole, and cross-stream regions. However, fully–coupled and elliptic simulation of this flowfield captured much of the flow physics since predictions effectively supplement, support, and help explain experimental data and observations from flow visualization tests.
- Distribution of dependent variables at the jet exit plane was highly two–dimensional and fundamentally different than those observed in large L/D situations. The final distribution resulted from the interaction of three competing mechanisms, namely, counter–rotating structure and local jetting effects within the film–hole, and crossflow blockage.
- Flow within the film–hole was shown to be extremely complex. It contains counter–rotating vortices and local jetting effects which make the flowfield in this region highly elliptic. Strength of these structures was found to be controlled mainly by a combination of three parameters, namely, film–hole length–to–diameter ratio, blowing ratio, and injection angle.
- The main source of turbulence shifted from the shear layer above the coolant jet in the cross–stream at low blowing ratios to the layer between the vortices and jetting regions within the film–hole at high blowing ratios.
- With proper care and an effective solution strategy, full convergence was attained for this strongly coupled and elliptic film cooling flowfield in a multi-zone domain at a blowing ratio as high as 2.0.
ACKNOWLEDGMENTS

The authors are indebted to Dr. Mikio Suo for his support and encouragement, and for his genuine interest in film cooling heat transfer. We would like to thank Profs. M.E. Crawford and D.G. Bogard for providing feedback on our predictions. We also appreciate fruitful discussions with our colleagues Drs. B.K. Sultanian and C. Prakash. Another colleague, Dr. R.K. Rout, provided expert advice on the use of I-DEAS. Last but not least, the authors wish to thank GE Aircraft Engines for permission to publish this paper.

REFERENCES


