Heat Transfer Engineering

Publication details, including instructions for authors and subscription information:
http://www.informaworld.com/smpp/title~content=t713723051

Reynolds Stress Transport Modeling of Film Cooling at the Leading Edge of a Symmetrical Turbine Blade Model

Fadéla Nemdili a; Abbès Azzi a; Georgios Theodoridis b; Bassam A. Jubran c

a Laboratoire de Mécanique Appliquée, Faculté de Génie, Mécanique Université des Sciences et de la Technologie d'Oran, Oran, Algeria

b Flow Dynamics Hellas S.A. Simulation-Measurement-Optimization-Control, Thessaloniki, Greece

c Department of Aerospace Engineering, Ryerson University, Toronto, Ontario, Canada

Online Publication Date: 01 November 2008

To cite this Article: Nemdili, Fadéla, Azzi, Abbès, Theodoridis, Georgios and Jubran, Bassam A. (2008) 'Reynolds Stress Transport Modeling of Film Cooling at the Leading Edge of a Symmetrical Turbine Blade Model', Heat Transfer Engineering, 29:11, 950 — 960

To link to this article: DOI: 10.1080/01457630802186064
URL: http://dx.doi.org/10.1080/01457630802186064

PLEASE SCROLL DOWN FOR ARTICLE

Full terms and conditions of use: http://www.informaworld.com/terms-and-conditions-of-access.pdf

This article maybe used for research, teaching and private study purposes. Any substantial or systematic reproduction, re-distribution, re-selling, loan or sub-licensing, systematic supply or distribution in any form to anyone is expressly forbidden.

The publisher does not give any warranty express or implied or make any representation that the contents will be complete or accurate or up to date. The accuracy of any instructions, formulae and drug doses should be independently verified with primary sources. The publisher shall not be liable for any loss, actions, claims, proceedings, demand or costs or damages whatsoever or howsoever caused arising directly or indirectly in connection with or arising out of the use of this material.
Reynolds Stress Transport Modeling of Film Cooling at the Leading Edge of a Symmetrical Turbine Blade Model

FADÉLA NEMDILI,1 ABBÈS AZZI,1 GEORGIOS THEODORIDIS,2 and BASSAM A. JUBRAN3
1Laboratoire de Mécanique Appliquée, Faculté de Génie, Mécanique Université des Sciences et de la Technologie d’Oran, Oran, Algeria
2Flow Dynamics Hellas S.A. Simulation-Measurement-Optimization-Control, Thessaloniki, Greece
3Department of Aerospace Engineering, Ryerson University, Toronto, Ontario, Canada

This paper investigates the performance of the SSG (Speziale, Sarkar, and Gatski) Reynolds Stress Model for the prediction of film cooling at the leading edge of a symmetrical turbine blade model using the CFX 5.7.1 package from ANSYS, Inc. Using a finite-volume method, the performance of the selected turbulence model is compared to that of the standard k−ε model. The test case blade model is symmetric and has one injection row of discrete cylindrical holes on each side near the leading edge. Numerical simulations are conducted for three different blowing ratios; film cooling effectiveness contours on the blade surface and lateral averaged adiabatic film cooling effectiveness are presented and compared with available measurements. The computations with the standard k−ε model reproduce the well-known underpredicted lateral spreading of the jet, and, consequently, lower values of the lateral averaged adiabatic film cooling effectiveness has been obtained. On the other hand, the second order Reynolds Stress Model yields reasonably good agreement with measurement data. In addition to validation data, several longitudinal and transversal contours and vector planes are reproduced and clearly underscore the anisotropic turbulent field occurring in the present shower head film cooling configuration.

INTRODUCTION

According to the thermodynamic cycle, it is well established that specific power as well as specific fuel consumption of gas turbine engines can be improved by increasing the turbine inlet temperature. Consequently, an efficient cooling system is a key factor to achieve high performance and increase the life cycle of gas turbine components. Blades in the first stages of gas turbines are by far the most exposed part. More precisely, the stagnation region near the leading edge is highly exposed to the hot free stream gases, and it is considered to be the most critical part in the blade where advanced stagnation cooling techniques such as showerhead film cooling injection are applied. Compared to the flat plate injection, the stagnation film cooling is more complex due to additional effects of thin boundary layer, high acceleration, and a nearly opposite coolant direction to the main stream. It is well known that the flow structure in the vicinity of the film cooling discharge holes is extremely complex and leads to several vortex structures. Previously, Bergeles et al. [1] and then Andreopoulos and Rodi [2] studied in detail a single perpendicular jet in cross-flow and highlighted the importance of the jet deflection toward the wall, which is associated by lower pressure in the wake and higher pressure in the front of the jet. The low pressure region in the wake is responsible for a flow motion toward the center plane, which promotes the formation of the two counter-rotating vortexes. Additional vortex structures are detected, such as the horseshoe, windward, and lee vortex. It is expected that the origin of the longitudinal vortex is inside the jet hole. Therefore, good prediction of the flow and cooling effectiveness depends crucially on how much the computational grid is refined in the vicinity of the jet.

From several previous studies, it is well argued that a jet in cross-flow is highly influenced by several parameters. Many experimental as well as numerical investigations are devoted to understanding the influence of parameters, such as wall curvature; free stream turbulence; compressibility; flow unsteadiness;
hole size, shape, and location; and angle of injection, among many others. Bogard and Thole [3] have just published a review paper on film cooling process and the related parameters that affect its performance. They supported Bunker [4] in that the bulk of the research has used a simple flat plate to study film cooling, which may be different from actual operating conditions of gas turbine blades. Some of the studies are devoted to leading edge region, which is represented by a semi-cylinder followed by a flat plate [5–7]. Recently, full blade models have been investigated numerically [8]. The test case used in the present numerical investigation deals with a shower head film cooling on a symmetrical blade, for which experimental data are provided by Haslinger and Hennecke [5]. In a previous numerical investigation, Lakehal et al. [9] presented a numerical investigation of the same symmetrical blade at that of [5] but with lateral injection. The performance of a new class of two-layer turbulence model combined with non-isotropic representation of the turbulent transport was tested and found to predict correctly the spanwise spreading of the temperature field and reduce the strength of the secondary vortices. Several versions of quadratic and cubic algebraic stress models were then tested for the same blade by Azzi and Lakehal [8].

From a numerical modeling point of view, it is well established that isotropic eddy viscosity turbulence models are not suitable for reasonably reproducing such complex cases. Several numerical investigations, including those of Azzi and Lakehal [10], have shown that two-equation models like the universal $k−\varepsilon$ model underpredict the lateral spreading of the temperature field and consequently lead to lower values of the laterally averaged adiabatic film cooling effectiveness. Second-order closure Reynolds stress models do not use the eddy viscosity hypothesis; instead, they solve a transport equation for each stress component. This class of model is the most sophisticated turbulence model available in the Reynolds Averaged Navier Stokes (RANS) strategy and is used in the present work. Andreini et al. [11] and Plesniak [12] predicted film cooling using $k−\varepsilon$, $k\omega$, and the Reynolds Stress Turbulence Model (RSM). They reported that the RSM model did not result in a significant improvement over the eddy viscosity model, which agrees with the findings of Walters and Leylek [13] and Azzi and Lakehal [10]. Holloway et al. [14] used the unsteady turbulence model to predict film cooling. This model improved the prediction of fundamental physics of film cooling by modifying the traditional turbulence model to account for the unsteadiness in the film cooling process.

On the other hand, numerical prediction is known to be conditioned by the near wall modeling strategy adopted. In the present computation, a scalable wall function feature is used [15]. The scalable wall function is a technique used in CFX [16] to enable the computational grid to be highly refined near the solid wall and still use the logarithmic wall function. This is accomplished by assuming that the solid surface coincides with the edge of the viscous sub-layer. Then, the computed non-dimensional wall distance is not allowed to fall below the limiting value that defines the width of the viscous sub-layer. The main advantage from that is the possibility to use highly near-wall grid refinement and still get improved prediction. This technique is documented and highly tested by CFX developer group [16].

**TURBULENCE MODEL**

The present simulations were conducted using the CFX 5.7.1. In the solver package, the solution of the RANS and energy equations is obtained using the finite volume method with a body-fitted hexahedral unstructured grid. A co-located layout is employed in which the pressure, turbulence, and velocity unknowns share the same location. The momentum and continuity equations are coupled through a pressure correction scheme, and several implicit first- and second-order accurate schemes are implemented for the space and time discretizations. In the present computation, convection terms are discretized with a second-order scheme except near discontinuities, where it reduces to first order to preserve boundedness.

Two different turbulence closures are evaluated in this paper. The first one, which will be used as a reference for the current industrial state of the art, is the isotropic eddy-viscosity $k−\varepsilon$ model [17]. This model is known to provide good predictions for many flows of engineering interest with some limitations for complex configurations, such as the boundary layer separation, swirling flows, and over curved surfaces. The second model is based on transport equations for the individual components of the Reynolds stress tensor and the dissipation rate. Theoretically, due to the anisotropic property, the Reynolds stress model is expected to be more universal and suitable for complex flows. Nevertheless, it is mathematically more complex and consequently less robust and more time consuming. Among different variants available in the code, the more accurate one is the Speziale, Sarkar and Gatski-Reynolds Stress Model (SSG-RSM) [18], which is used in the present investigation. Compared to the $k−\varepsilon$ model, the Reynolds stress model has six additional transport equations to be solved for each time step.

The equation for the transport of the Reynolds stress is written in compact form:

$$
\frac{\partial}{\partial t}(\rho \overline{u_i u_j}) + \frac{\partial}{\partial x_k}(\rho \overline{u_i u_j} U_k) = P_{ij} + \Phi_{ij}$$

$$+ \frac{\partial}{\partial x_k} \left[ \left( \mu + \frac{2}{3} \frac{k^2}{\varepsilon} \right) \frac{\partial \overline{u_i u_j}}{\partial x_k} \right] - \frac{2}{3} \delta_{ij} \rho \varepsilon \quad (1)
$$

where $\Phi_{ij}$ is the pressure-strain correlation and $P_{ij}$ the exact production term.

As the turbulence dissipation appears in the individual stress equations, an equation for $\varepsilon$ is still required.

$$
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial}{\partial x_k} (\rho U_k \varepsilon) = \varepsilon \left( c_{11} \rho - c_{12} \rho \varepsilon \right)
$$

$$+ \frac{\partial}{\partial x_k} \left[ \left( \mu + \frac{\mu}{\sigma_{\varepsilon}} \right) \frac{\partial \varepsilon}{\partial x_k} \right] \quad (2)
$$
The most important term in Reynolds stress models is the pressure-strain correlation $\Phi_{ij}$, which can be expressed in the general form by:

$$\Phi_{ij} = \Phi_{ij1} + \Phi_{ij2}$$  \hspace{1cm} (3)

where

$$\Phi_{ij1} = -\rho \varepsilon \left( C_{s1} a + C_{s2} \left( aa - \frac{1}{3} a \cdot a \delta \right) \right)$$  \hspace{1cm} (4)

$$\Phi_{ij2} = -C_{r1} P a + C_{r2} \rho k S - C_{r3} \rho k S \sqrt{a \cdot a}$$

$$\frac{a}{\rho} \left( \frac{a S^T + S a^T}{2} - \frac{2}{3} a \cdot S \delta \right) + C_{r5} \rho k \left( a W^T + W a^T \right)$$

$$a = \frac{u \otimes u}{k} - \frac{2}{3} \delta$$  \hspace{1cm} (6)

$$S = \frac{1}{2} \left( \nabla U + (\nabla U)^T \right)$$  \hspace{1cm} (7)

$$W = \frac{1}{2} \left( \nabla U - (\nabla U)^T \right)$$  \hspace{1cm} (8)

In this formulation, $a$ is the anisotropy tensor, $S$ is the strain rate and $W$ is the rotation rate tensor. This general form can be used to model linear and quadratic correlations by using appropriate values for the constants.

The SSG version used in the present investigation was developed by Speziale et al. [18] and uses a quadratic relation for the pressure-strain correlation. The model constants are listed in Table 1.

### NEAR WALL TREATMENT

The major discrepancy of the standard wall function approach is its dependence on the nearest point to the wall. It is shown that refining the near wall mesh does not give a unique solution of increasing accuracy [15]. The problem of inconsistencies in the wall-function in the case of fine meshes is overcome with the use of the Scalable Wall Function formulation developed by the CFX group. The basic idea behind the scalable wall-function approach is to limit the value used in the logarithmic formulation by a lower value of $\tilde{y}^+$, which is the intersection between the logarithmic and the linear near wall profile. The computed $\tilde{y}^+$ is therefore not allowed to fall below this limit. Thus, all mesh points are outside the viscous sublayer, and all fine mesh inconsistencies are avoided. This approach allows the use of scalable wall function on arbitrarily fine meshes.

A second improvement added in the CFX formulation is the use of an alternative velocity scale, $u^*$, in the logarithmic region:

$$u^* = C \sqrt[4]{k} / l^{1/2}$$  \hspace{1cm} (9)

As it is well known in turbulent flow $k$ never goes to zero value, thus $u^*$ does not go to zero if $U_\infty$ goes to zero. Based on this definition, the following explicit equation for $u^*$ can be obtained:

$$u_* = \frac{U_i}{l} \log(\tilde{y}^+ / C)$$  \hspace{1cm} (10)

The absolute value of the wall shear stress $\tau_\omega$, is then obtained from:

$$\tau_\omega = \rho u^* u_*$$  \hspace{1cm} (11)

### TURBULENT HEAT FLUX

The turbulent heat flux is modeled by use of the eddy diffusivity hypothesis, where by analogy to eddy viscosity hypothesis the Reynolds fluxes of a scalar are linearly related to the mean scalar gradient. Eddy diffusivity is related to eddy viscosity by use of a turbulent Prandtl number.

### TEST CASE DESCRIPTION

The blade used in the present study is symmetrical with a length of 515 mm and a maximum width of 72 mm, as seen in Haslinger and Hennecke [5]. The leading edge of the model has one row of holes on each side ($D = 4$ mm) with a lateral spacing of 5D. In the streamwise direction, the holes are inclined 110˚ to the surface and located so that the trailing edge of the hole is at $s/D = 3.1$, where $s$ is the length along the blade from the stagnation point and $s/D = 0$ is the blade stagnation line. The geometry is shown in Figure 1. Two configurations with different inclinations of the injection holes were investigated in the experimental study: one without lateral inclination, $\gamma = 0^\circ$ (streamwise injection) and one with a lateral inclination of $\gamma = 45^\circ$ (lateral injection). The approach-flow velocities were in the range ($U = 15–30$ m/s) so that the flow can be considered incompressible. The free-stream turbulence level was below 0.5%. In the present investigation, calculations were carried out only for the streamwise injection and an approach velocity of 30 m/s. The length to diameter ratio of the film holes is 4 and the density ratio is 1.

Three blowing ratios ($M = \rho U_1/\rho U_∞ = 0.3, 0.5,$ and 0.7) are considered and compared to the experimental ones. Computations with higher blowing ratio than $M = 0.7$ showed numerical instability and appear to be indicating that the physics of the

---

Table 1. The SSG-RSM model constants

<table>
<thead>
<tr>
<th>$c_4$</th>
<th>$c_{r1}$</th>
<th>$C_{s1}$</th>
<th>$C_{s2}$</th>
<th>$C_{r1}$</th>
<th>$C_{r2}$</th>
<th>$C_{r3}$</th>
<th>$C_{r4}$</th>
<th>$C_{r5}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.22</td>
<td>1.45</td>
<td>1.83</td>
<td>1.7</td>
<td>-1.05</td>
<td>0.9</td>
<td>0.8</td>
<td>0.65</td>
<td>0.625</td>
</tr>
</tbody>
</table>

vol. 29 no. 11 2008
problem turns to be unsteady. In this case, further computations with unsteady RANS or Large Eddy Simulation methods seem to be appropriate for obtaining more realistic results [19].

**COMPUTATIONAL DOMAIN**

Due to the streamwise injection, only half of the flow domain is considered in the numerical computation. The domain extends from a plane passing through the injection hole to a middle plane between two holes, and symmetry conditions are imposed on those planes. Lateral direction is defined as perpendicular to the two planes. The upper boundary is placed $45D$ over the symmetry plane at the location of the wind tunnel wall. The inflow boundary is located $90D$ upstream of the leading edge and the outflow boundary $96D$ downstream of the trailing edge. In order to be more realistic on injection boundary condition, the injection tube was included in the computational domain. This is a crucial condition, especially for low blowing ratio, where the velocity profile at the hole boundary cannot be set up beforehand [4, 20]. The computational extent of the domain is shown in Figure 2.

**BOUNDARY CONDITIONS**

A uniform streamwise velocity profile was applied $U_\infty = 30$ m/s at the inflow boundary; the Reynolds number based on $D$ and $U_\infty$ was $Re = 7950$. Uniform distributions were also specified for $k$ and $\varepsilon$, corresponding to a turbulence intensity of $Tu = 0.5\%$ and $\mu_t/\mu = 30$. Similarly, a uniform velocity profile was set at the inlet of the discharge pipe. Here also, uniform distributions of $k$ and $\varepsilon$ were specified, based on $Tu = 3\%$ and a length scale of $k^{3/2}/\varepsilon = 0.3D$. The injection flow velocity was computed according to the blowing ratio values ($M = \rho_cU_c/\rho_\infty U_\infty$). The outflow boundary condition was set at constant zero relative pressure, while on the blade surface and the pipe internal walls, adiabatic wall, $k = 0$, and no-slip conditions were employed.

**COMPUTATIONAL GRID**

Preliminary calculations for the lower blowing ratio ($M = 0.3$) were performed, and lateral averaged film cooling effectiveness was compared in order to establish the grid size that will be used in the next computations. Three levels of mesh refinement were used, which consisted of approximately 250,000, 500,000, and 1,000,000 hexahedral elements. The final adopted grid was composed by nearly 1,000,000 hexahedral cells and 16 blocks. The distribution of the computational nodes was highly refined near the blade. Due to the importance of the region in the vicinity of the hole injection, the computational grid was highly refined in this place. On the other hand, the quality of the computational grid is highly improved in the sense of aspect ratio and skewness by use of the well known O grid strategy [21]. Figure 3 shows the zoomed region of the computational grid in the vicinity of the injection hole, which was created using the ICEM CFD 10.0 grid generation tool from ANSYS, Inc., version 10.0 [19]. In order to be sure that convergence is reached, the target value of the rms residuals , which is set by default to $10^{-4}$, was switched here to $10^{-6}$.
RESULTS AND DISCUSSION

A so-called laterally averaged adiabatic film cooling effectiveness \(<\eta>\) is defined by:

\[
<\eta> = \frac{1}{L} \int_{L} \eta \, dz
\]

(12)

where \(L\) represents the spanwise dimension of the blade and \(\eta\) is the effectiveness, defined by:

\[
\eta = \frac{T_{\infty} - T}{T_{\infty} - T_{c}}
\]

(13)

Longitudinal distribution of the laterally averaged effectiveness \(<\eta>\) computed with the standard \(k-\varepsilon\) turbulence model (SKE) is compared to available measurements for the three blowing ratios. The two vertical lines show the hole position on the blade, while \(s/D = 0\) is the blade stagnation line (see Figure 4).

As predicted by previous numerical investigations, the standard \(k-\varepsilon\) model under-predicts the lateral expansion of the thermal field and consequently under-predicts the laterally averaged adiabatic film cooling effectiveness. For \(M = 0.3\), results with grids 2 and grid 3 are collapsed until \(s/D = 5\), and then a difference of approximately 33\% is marked at \(s/D = 25\). The effect on the higher blowing ratio is less pronounced when it is more pronounced for the intermediate blowing ratio. The reason is probably due to the extreme complexity for the intermediate case, as for the higher blowing ratio, the jet lift off far from the wall and the near wall flow is less disturbed than it is for the intermediate blowing ratio.

Although the difference of results between grids 2 and 3 indicates that there is a grid dependency for \(s/D > 5\), a more refined mesh was not tested, as grid 3 was the biggest mesh that could run on the available hardware. Thus, grid 3 was adopted in all subsequent simulations.

Figure 5 shows the laterally averaged effectiveness computed using grid G3 with \(k-\varepsilon\) and RSM (SSG) turbulence models. Comparison with experimental data has lead to the following observations:

- For the lower blowing ratio injection, the jet is kept close to the wall, and the highest value of \(<\eta>\) is occurring near the trailing edge of the injection hole.
- It then decreases monotonically and rapidly until \(s/D = 5\), and then it decreases with a smaller gradient due to the mixing with the main stream.
- For the two higher blowing ratios, experimental data show that the jet lift off and reattach at approximately \(s/D = 5\).

The trend of this variation is more or less captured by the two turbulence models used, but with some differences. As it is expected and shown previously, the first order isotropic \(k-\varepsilon\) turbulence model under-predicts the laterally averaged effectiveness. At least for the lower blowing ratio, the second order Reynolds stress model RSM (SSG) is superior in capturing the experimental results with a slight over-prediction in vicinity of heat transfer engineering...
Figure 5 Laterally averaged adiabatic film cooling effectiveness computed on grid G3, with $k-\varepsilon$ and RSM (SSG) turbulence models: (a) $M = 0.3$, (b) $M = 0.5$, and (c) $M = 0.7$.

$s/D = 5$ point, which is the region where the $\langle \eta \rangle$ decrease changes its rate. For the two higher blowing ratios, the trend is well captured by the RSM model but with under-prediction, especially after $s/D = 5$ position. Nevertheless, for the higher

Figure 6 Laterally averaged adiabatic film cooling effectiveness computed with RSM (SSG) turbulence model: $M = 0.3$, $0.5$, and $0.7$.

Figure 7 Contours of adiabatic film cooling effectiveness with RSM (SSG) model (left panel) and SKE model (right panel) and for $M = 0.3$, $0.5$, and $0.7$. Contours are from 0.1 to 0.9 with an increment of 0.1.
blowing ratio, the level of the SKE results is in good agreement with experiment after the $s/D = 7$ position.

Laterally averaged adiabatic film cooling effectiveness for three blowing ratios ($M = 0.3, 0.5, \text{ and } 0.7$) are presented together with experimental measurements in Figure 6. Except for the lowest blowing ratio ($M = 0.3$), which is well simulated, the other results are mainly under-predicted. Nevertheless, the trend of variation is well reproduced, especially the fact that the effectiveness decreases monotonically in the vicinity of $s/D = 5$ and $s/D = 6.5$ in computational results and of $s/D = 4.5$ in the experimental results. After that, the trend is inverted, and increasing variation of $\langle \eta \rangle$ is observed. This behavior indicates that the jet lifts off from the wall in the vicinity of the injection hole and then reattaches afterward. Far downstream and at approximately $s/D = 25$, the same value of $\langle \eta \rangle$ is reached for all cases. This is due mainly to the mixing with the mainstream flow.

The distributions of adiabatic film cooling effectiveness on the blade are presented on Figure 7 for the different blowing ratios. Nine effectiveness contours beginning with 0.1 until 0.9 are plotted on each figure. A common feature of all sub-figures is that in the region directly above the hole, the effectiveness is lower than unity. This is an indication that in this part of the flow, the heat diffusion overcomes the convection. This is related directly to the level of the blowing ratio. Thus, for the lowest injections rate, this zone occurs in approximately the first half of the hole, while for the highest one it is much smaller. Results for the lowest blowing ratio with the SKE turbulence model are reported on the left panel in order to be compared with those obtained with the RSM model. When looking at the centerline distributions, it is obvious that the results with the SKE model decrease rapidly to very low values, while those with RSM model are maintained at higher values far downstream. This distribution explains the higher distribution of the lateral averaged adiabatic film cooling effectiveness mentioned in the previous figures. When increasing the blowing ratio, the jet lift off and lower film cooling effectiveness values are reported downstream of the hole injection. Velocity vectors at the symmetry plan for $M = 0.3$ and $M = 0.7$ are presented in Figure 8. The jet flow interacts strongly with the main flow and is then deflected in the streamwise direction. A small zone of low fluid velocity is then formed downstream of the trailing edge of the hole. The size of this zone increases when the blowing ratio increases and passes from approximately one hole diameter to two for $M = 0.3$ and $M = 0.7$, respectively. In this zone, the flow is mainly coming from lateral sides, as will be shown later. The penetration of the jet in the main flow is also related to the blowing ratio and passes from slightly less than one hole diameter to more than one diameter for $M = 0.3$ and $M = 0.7$, respectively. The corresponding contours of film cooling effectiveness are plotted on Figure 9 and show that the cooling zone for the lowest blowing ratio ($M = 0.3$) is two times thinner than it is for the highest blowing ratio ($M = 0.7$). Thus, for high blowing ratio, the jet flow is strongly mixed with the main flow, and consequently the cooling effectiveness drops down compared to that for the low blowing ratio situation. As was mentioned previously, in a zone of approximately half-hole diameter and where there is a heat diffusion in such a low flow speed area, the effectiveness is less than unity. This is true for $M = 0.3$, and the zone is much smaller than that for the highest blowing ratio. This supports the argument to include the hole in the computational domain.

The structure of the fluid flow and the thermal field in planes perpendicular to the blade surface and at constant $s/D$ are presented in Figures 10 and 11 for $M = 0.3$ and $M = 0.7$, respectively. The planes are at $s/D = 3.5, 6.5, \text{ and } 10$, respectively, and are chosen such that the first one is a half hole diameter downstream of the trailing edge of the injection hole. The second one
is expected to be in the low and reverse flow zone, and the third is far downstream in the reattached zone. The flow characteristics of two counter-rotating vortices and the kidney shape are well reproduced in the first plane and are still present in the other two planes. For $M = 0.3$, the two vortices are close to the wall and indicate that the flow enters laterally into the reverse flow region. Further downstream ($s/D = 6.5$ and 10), the counter-rotating vortices are still present, but much weaker, and lose its original kidney shape (see Figure 10). The situation is different for the high blowing ratio ($M = 0.7$). Figure 11 clearly shows that the

**Figure 10** Velocity vectors and effectiveness contours on three planes perpendicular to the blade, $M = 0.3$ (contours values are from 0.1 to 0.9 with a step of 0.1).

**Figure 11** Velocity vectors and effectiveness contours on three planes perpendicular to the blade, $M = 0.7$ (contours values are from 0.1 to 0.9 with a step of 0.1).
two counter-rotating vortices are much stronger and still present in all planes. These vortices are known to be responsible for draining the hot main stream flow toward the wall. The decrease of cooling effectiveness is strongly related to this phenomenon.

In order to give an idea of the topology of the mean velocity field in the vicinity of the injection hole, streamlines are plotted on Figure 12 for $M = 0.3$ and $M = 0.7$. Streamlines issuing from the hole injection and others from the mainstream flow are plotted with different thickness. For clarity of the figures, other streamlines that turn on recirculating flow are not presented here. The formation of the secondary flow is clearly illustrated by the trajectory of the streamlines issuing from the downstream half of the hole. One can see that the flow is immediately directed in the lateral direction and meeting each other in the symmetry plan, and then it goes up in the vertical direction. This phenomenon is more pronounced for $M = 0.7$ than it is for $M = 0.3$.

**CONCLUSIONS**

Computational prediction of showerhead film cooling with streamwise injection from one hole row in each side of symmetrical blade model is presented. The known lack of the first order isotropic $k-\epsilon$ turbulence model to reproduce the thermal and flow field is overcome by the use of a second order Reynolds Stress Model. By resolving six more transport equations, which is of course more time consuming, the model is more suited to reproduce the anisotropic feature of the flow and take into account the curvature effects. Results are successfully compared to the available experimental measurements, and effects of different blowing ratios are highlighted. As many researchers have found, increasing blowing ratio negatively affects the film cooling effectiveness distributions on the blade. Laterally averaged film cooling effectiveness for the lowest blowing ratio ($M = 0.3$) is accurately reproduced by the model, while other cases are less satisfactorily captured. Computations with higher blowing ratio than $M = 0.7$ showed numerical instability and appear to indicate that the physics of the problem turns to be unsteady. In this case, further computations with Unsteady-RANS or Large Eddy Simulation methods seem to be appropriate to produce more realistic results [20]. To date, there have been rather limited studies on the use of advanced turbulence models (LES and DES) to predict film cooling, where most of the works reported in the literature were related to a single jet in cross-flow or simple geometry hole. However, there are encouraging signs regarding the use of such models to accurately predict and capture the thermal and fluid flows of the film cooling process. The challenge would be how to model the unfiltered fields that result in subgrid scale stresses.

**NOMENCLATURE**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$a$</td>
<td>anisotropy tensor</td>
</tr>
<tr>
<td>$C$</td>
<td>log layer constant in Eq. (10)</td>
</tr>
<tr>
<td>$C_{r_1}, C_{r_2}, C_{r_3}, C_{r_4}$</td>
<td>turbulence model constants</td>
</tr>
<tr>
<td>$c_1, c_2$</td>
<td>turbulence model constants</td>
</tr>
<tr>
<td>$C_{s_1}, C_{s_2}, C_{s_5}$</td>
<td>turbulence model constants</td>
</tr>
<tr>
<td>$C_{\mu}$</td>
<td>model constant in Eq. (9)</td>
</tr>
<tr>
<td>$D$</td>
<td>hole diameter</td>
</tr>
<tr>
<td>$k$</td>
<td>turbulence kinetic energy</td>
</tr>
<tr>
<td>$L$</td>
<td>spanwise dimension of the blade</td>
</tr>
<tr>
<td>$M$</td>
<td>blowing ratio</td>
</tr>
<tr>
<td>$P_{ij}$</td>
<td>production term</td>
</tr>
<tr>
<td>$S$</td>
<td>strain rate</td>
</tr>
<tr>
<td>$s$</td>
<td>length along blade from the stagnation point</td>
</tr>
<tr>
<td>$T$</td>
<td>temperature</td>
</tr>
<tr>
<td>$T_u$</td>
<td>turbulence intensity</td>
</tr>
<tr>
<td>$U$</td>
<td>velocity</td>
</tr>
<tr>
<td>$u_i, u_j$</td>
<td>Reynolds stresses components</td>
</tr>
<tr>
<td>$U_r$</td>
<td>velocity tangent to the wall</td>
</tr>
<tr>
<td>$u_e$</td>
<td>friction velocity</td>
</tr>
<tr>
<td>$u^*$</td>
<td>alternative velocity scale</td>
</tr>
<tr>
<td>$W$</td>
<td>rotation rate tensor</td>
</tr>
<tr>
<td>$y^+$</td>
<td>dimensionless distance from the wall</td>
</tr>
<tr>
<td>$\tilde{y}^+$</td>
<td>scalable dimensionless distance from the wall</td>
</tr>
<tr>
<td>$z$</td>
<td>spanwise direction</td>
</tr>
</tbody>
</table>

**Greek Symbols**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\delta$</td>
<td>boundary layer thickness</td>
</tr>
<tr>
<td>$\delta_{ij}$</td>
<td>Kronecker symbol</td>
</tr>
</tbody>
</table>
References


Fadela Nemdili is a Ph.D. student at the Université des Sciences et de la Technologie d’Oran, Oran, Algeria. She received her master’s degree from the ENSET School, Algeria, in 2003. She is working on film cooling computations.

Abbès Azzi is a mechanical engineer with a background of computational heat transfer and turbulence modeling. He received his Ph.D. in 2002 from the Université des Sciences et de la Technologie d’Oran (USTO), Oran, Algeria, working on film cooling computation in collaboration with ETH Zurich. He spent one year as a post-doctoral researcher with the group of Pr. Lesieur at Grenoble, France, working on large eddy simulation. Currently, he is an associate professor at USTO.
Georgios Theodoridis is a mechanical engineer and has a Ph.D. from Aristotle University, Thessaloniki, Greece, in 1993. He is currently working with Flow Dynamics Hellas S.A, Thessaloniki, Greece. He has participated in the development of advanced software in the fields of computational fluid dynamics and turbulence modeling. He has performed computational work in numerous fields, the most important among them turbine blade film cooling, wind turbines, wind loading, compressible flows, dispersion and chemical transformation of pollutants in urban areas, and two-phase flows in secondary clarifiers of wastewater treatment plants, wax coating and water mineral oil separation. He is an author or co-author of more than 30 refereed scientific publications.

Bassam A. Jubran is a professor of thermo-fluid engineering. He was educated at Cardiff University (formerly University of Wales), UK, graduating in 1980 with B.Sc. Honors degree in mechanical engineering. He obtained his Ph.D., also from the University of Wales, in 1984. Professor Jubran joined the Department of Aerospace Engineering at Ryerson University, Toronto, Canada, in 2004. He served as editor-in-chief of the Journal of Engineering Research, and he is now on the editorial board of the new International Journal of Low Carbon Technologies, published by Manchester University Press, UK. He has published more than 120 papers in top international peer journals and international conference proceedings.