JOURNAL OF THERMOPHYSICS AND HEAT TRANSFER
Vol. 20, No. 4, October–December 2006

# Louver Cooling Scheme for Gas Turbines: Multiple Rows

X. Z. Zhang* and I. Hassan†
*Concordia University, Montréal, Québec H3G 1M8, Canada*
and
T. Lucas
*Pratt and Whitney Canada, Longueuil, Québec J4G 1A1, Canada*

DOI: 10.2514/1.20599

**The arrangements of two and three rows of holes of a new advanced cooling scheme for gas turbine blades have been studied numerically in this paper and their performance compared with other cooling schemes such as traditional circular holes and discrete slots. The new hole was designed in such a way that the coolant must go through a bend before exiting the blade, thus impinging on the blade material. The flared hole exit was also designed to reduce the coolant momentum in the normal direction and to ensure wide lateral spreading of the coolant on the downstream surface, making the use of coolant air more efficient. Turbulence was modeled using realizable $k$–$\epsilon$ turbulence model. Three configurations have been studied: two rows of inline holes, two rows of staggered holes, and three rows of staggered holes with holes equally distanced in the span-wise direction. The new scheme produces the highest cooling effectiveness and lowest heat transfer levels among the cooling schemes compared since jet liftoff is avoided. The staggered arrangement offers a much higher laterally averaged cooling effectiveness with a slightly lower heat transfer coefficient than its inline arrangement counterpart.**

## Nomenclature

| | | |
|---|---|---|
| DR | = | density ratio, $\rho_j/\rho_\infty$ |
| $d$ | = | hydraulic diameter of the hole inlet, m |
| $E$ | = | total energy, J |
| $h$ | = | heat transfer coefficient, $W/m^2 \cdot K$ |
| $k$ | = | turbulent kinetic energy, $m^2/s^2$ |
| $m$ | = | blowing ratio (or rate), $\rho_j U_j/\rho_\infty U_\infty$ |
| $P$ | = | pressure, Pa |
| $p$ | = | pitch of holes: center-to-center hole spacing, m |
| $q''$ | = | surface heat flux per unit area, $W/m^2$ |
| $Re_d$ | = | Reynolds number based on freestream velocity and injection diameter, $\rho_\infty U_\infty d/\mu_\infty$ |
| $s$ | = | row spacing, m |
| $T$ | = | temperature, K |
| $U$ | = | velocity, m/s |
| $u$ | = | streamwise velocity component, m/s |
| $\boldsymbol{v}$ | = | velocity vector, m/s |
| $x$ | = | streamwise coordinate, m |
| $y$ | = | vertical coordinate, m |
| $z$ | = | spanwise coordinate, m |
| $\epsilon$ | = | dissipation rate of turbulent kinetic energy, $m^2/s^3$ |
| $\mu$ | = | dynamic (laminar) viscosity, $kg/m \cdot s$ |
| $\rho$ | = | density, $kg/m^3$ |
| $\underline{\tau}$ | = | stress tensor, Pa |

*Subscripts and Superscripts*

| | | |
|---|---|---|
| aw | = | adiabatic wall |
| $f$ | = | conditions with film cooling |
| $j$ | = | refers to the jet |
| $o$ | = | conditions in the absence of film cooling |
| $w$ | = | wall conditions |
| $\infty$ | = | mainstream conditions at inlet plane and in freestream |

## I. Introduction

THE efficiency of gas turbines is dependent on a number of factors, among which the turbine inlet temperature is of significant importance. Although advanced alloys allow higher turbine inlet temperatures, improvements in metallurgy since 1960 have been largely stalled with a 60°C increase over the last four decades, or roughly 1.5°C per year. Moreover, the inlet temperatures of gas turbines have been continually rising approximately 20°C each year over the last three decades due to increasingly sophisticated cooling methods, such as film cooling, and this trend is unlikely to stop in the near future.

Film cooling has been heavily investigated during the last three decades. Publications on film cooling consist of nearly 2700 manuscripts (Bunker [1]), the majority of which have appeared in the period between 1970 and the present. Despite huge efforts to advance the understanding of this elusive problem, either experimentally or numerically, no significant progress, even at the most fundamental level, has been made. Only a few points are in consensus within the technical community. Most researchers have documented that decreasing the injection angle prevents jets from lifting off the protected surface, making the use of coolant more efficient. In addition, shaped holes with expanded exits reduce the momentum of coolant, and yield an increase in surface coverage.

The majority of studies focused on circular holes with simple or compound orientation angles. Only recently, in the last 10 to 15 years, have shaped holes received much attention. Schmidt et al. [2] and Thole et al. [3] documented that jets with expanded exits significantly reduce jet penetration and produce an improved lateral spread of the film-cooling jets. Compound angles lead to increased lateral spreading of injectant. Those findings were later confirmed by Bell et al. [4] and Taslim and Khanicheh [5]. Yu et al. [6] showed that shaped holes resulted in a lower and more uniform heat transfer coefficient as compared with cylindrical holes. In contrast, Sen et al. [7] reported significantly increased heat transfer levels at high blowing ratios in holes with a large compound angle, resulting in

Received 17 October 2005; revision received 21 February 2006; accepted for publication 14 March 2006. 

*Department of Mechanical and Industrial Engineering.
†Department of Mechanical and Industrial Engineering; IbrahimH@alcor.concordia.ca

poorer overall performance despite higher effectiveness. Lee et al. [8] recommended that the forward expanded hole be adopted in combination with a compound angle orientation as shaped holes with simple angles do not provide substantial improvement in film-cooling performance. Gritsch et al. [9] showed that laterally averaged film-cooling effectiveness was found to show only limited sensitivity to variations of hole geometry and that a compound angle has some detrimental effects at high blowing ratios.

Computational studies of shaped holes include Giebert et al. [10], Hyams and Leylek [11], and Bohn and Kusterer [12]. Giebert et al. [10] achieved satisfactory numerical results for shaped holes with forward-laterally expanded exits in terms of the adiabatic film-cooling effectiveness. This resulted from the absence of jet liftoff and subsequent reattachment. Hyams and Leylek [11] pointed out that forward diffused holes perform well along the centerline, but do not spread well laterally. Bohn and Kusterer [12] showed that fan-shaped configurations were up to three times more effective than cylindrical configurations.

Jubran and Maiteh [13], Ligrani et al. [14], Ahn et al. [15], Yuen and Martinez-Botas [16,17], and Maiteh and Jubran [18] studied two rows of circular holes. It was proved that two staggered rows of holes tend to provide better and more uniform cooling protection than two inline rows of holes. Ligrani et al. [14] found that compound angle injection configurations provided significantly improved protection as compared with simple angle configurations. Yuen and Martinez-Botas [16,17] concluded that staggered rows of holes improved the spanwise uniformity in effectiveness, but the heat transfer levels were larger than those of the inline rows with the same pitch. Dittmar et al. [19] illustrated that double rows of discrete slots provided better effectiveness at high blowing ratios than circular holes. Double row configurations also showed decreased heat transfer levels.

Numerical studies of shaped holes in multirow arrangements are rare. This is partly because the jet-in-a-cross phenomenon was not well understood, and partly because even with the most simple one circular jet numerical simulations failed to capture the jet liftoff effect in terms of cooling effectiveness at high blowing ratios (Leylek and Zerkle [20] and Walters and Leylek [21]). Consequently, the accuracy of the numerical results was left in doubt. In the current study, the performance of a new cooling scheme with two and three rows will be presented and compared with other cooling schemes. This will be the first time, to the best of the authors' knowledge, that an effort has been made to understand the performance of such a novel cooling scheme with arrangements consisting of more than one row of holes. With the concept of the proposed advanced cooling scheme, first designed by Immarigeon [22] in collaboration with Pratt and Whitney Canada and subsequently refined by the authors to show its full advantages, a greater portion of the airfoil is protected. The film hole was designed in such a way that the coolant must go through a bend before exiting the blade, as shown in Fig. 1, thus impinging on the blade material. Finally, the flow exits very close to the blade surface, minimizing aerodynamic losses.

## II. Governing Equations and Turbulence Modeling

The assumptions made for the flow were based on the test conditions of most experiments, and are as follows: 1) three-dimensional steady state, 2) incompressible, 3) viscous, 4) Newtonian flow, 5) turbulent, 6) single phase air, 7) no source of fluid or heat generation in the domain, and 8) negligible gravitational force. The flow is governed by the Navier–Stokes conservation equations, and under the preceding assumptions, the conservation equations of continuity, momentum, and energy become

Continuity:

$$\nabla \cdot (\rho \boldsymbol{v}) = 0 \tag{1}$$

Momentum:

$$\nabla \cdot (\rho \boldsymbol{v}\boldsymbol{v}) = -\nabla P + \nabla \cdot (\bar{\bar{\tau}}) \tag{2}$$

where

$$\bar{\bar{\tau}} = \mu\left[(\nabla \boldsymbol{v} + \nabla \boldsymbol{v}^T) - \tfrac{2}{3}\nabla \cdot \boldsymbol{v} I\right] \tag{3}$$

Energy:

$$\nabla \cdot [\boldsymbol{v}(\rho E + P)] = \nabla \cdot [k_{\mathrm{eff}} \nabla T + (\bar{\bar{\tau}}_{\mathrm{eff}} \cdot \boldsymbol{v})] \tag{4}$$

As turbulent flows are characterized by fluctuating fields of small scale and high frequency, which are computationally expensive, the instantaneous governing equations have been time-averaged to remove the small scales. As a result, new unknown variables have been created. These unknown variables have been defined differently in terms of known quantities, and have given rise to various turbulence models. When considering all types of problems, no single turbulence model is deemed superior over the others.

The literature survey indicates that the jet-in-cross flow is quite complicated. Consequently, the simulations usually do not agree very well with the experimental data. In many efforts to evaluate the performance of different turbulence models in film-cooling applications, numerical errors were usually too large to allow clear conclusions to be drawn. The mesh quality plays such an important role in determining the solution that the performance of the turbulence models could be completely masked. Lack of consistency is prevalent and the reliability of many methods in CFD analysis for film cooling is seriously in doubt. In this study, the realizable $k$–$\epsilon$ model (Shih et al. [23]), provided by the commercial software FLUENT, was selected to perform the simulations.

## III. Simulation Details

The new cooling scheme geometry is shown in Fig. 1. Three configurations of the new scheme were studied, namely, two rows of inline holes, two rows of staggered holes, and three rows of staggered holes equally distanced in the spanwise direction. The computational domain and boundary conditions were based on the assumption that the new scheme was tested in a wind tunnel with the characteristic length of $d = 12.7$ mm at low turbulence intensity of less than 3% and with a mainstream velocity of 20 m/s corresponding to Reynolds number of 16,000. This assumption was made only for comparison purposes, because in most of the experimental studies different cooling schemes were tested in a wind tunnel with low mainstream velocities. In the adiabatic cooling test the density ratio was assumed to be 2, whereas in the heat transfer test the density ratio was set at 1.

The computational domain includes one whole period from center to center. The details of the computational domain and boundary conditions can be found in Zhang and Hassan [24,25]. Different meshes were used at the beginning to determine the optimum grid

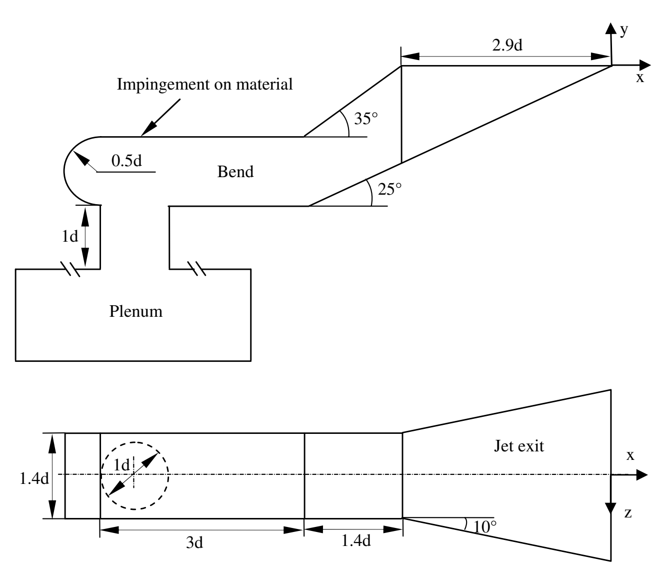



**Fig. 1 Geometry of the jet section of the new scheme.**

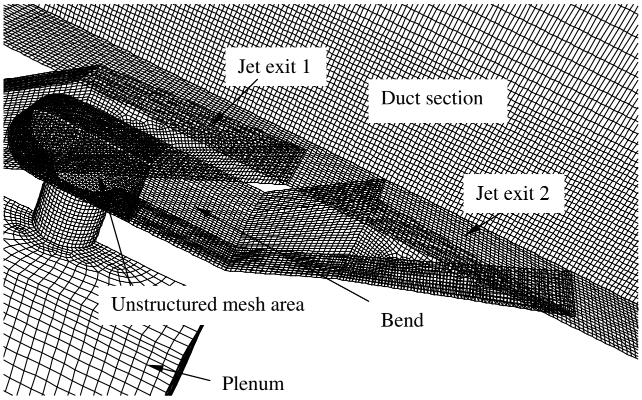



**Fig. 2 A typical mesh, multiblock structured (two staggered rows).**

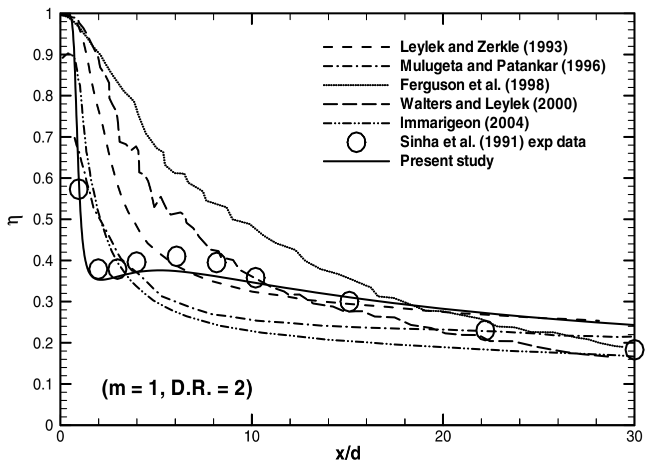



**Fig. 3 Present and previous predictions (Zhang and Hassan [24]).**

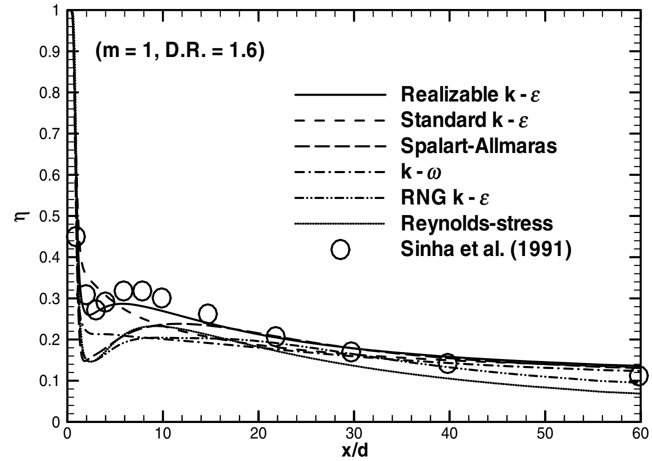



**Fig. 4 The performance of turbulence models (Zhang and Hassan [24]).**

size and to ensure a grid independent solution. The grids contained between $0.4 \times 10^6$ and $1 \times 10^6$ cells, which took roughly one full day of computing time to reach convergence using a workstation with a CPU speed of 3 GHz and a RAM of 1 GB. The previous study by Zhang and Hassan [24] has shown that a structured mesh yields more accurate predictions than its unstructured counterpart. In this study, as the geometry is more complicated than a simple circular jet, a multiblock structured mesh was used, therefore, structured mesh was created wherever possible. A typical mesh for the two rows of staggered holes arrangement is shown in Fig. 2. Because of the large number of cells, only half of the domain was shown to make the mesh structure more visible.

A methodology was developed to make the most of the current turbulence models in film-cooling simulations. Systematic benchmark studies were performed to find the most appropriate turbulence model, with the benchmark geometry close to the new scheme. High-quality structured meshes were created to fully resolve the features of the flowfield with most of the cells concentrated in areas of large variable gradients, because a hexahedron is more efficient to fill a volume than a tetrahedron. When unstructured meshes are used, it is very difficult to efficiently concentrate cells in the near-hole and wall regions to resolve the wake of recirculation and boundary layers. Particular attention was paid to the boundary layers. The present Reynolds-averaged turbulence models are empirical, or semi-empirical, and are only valid for the turbulence core flow far away from the walls. To render these models suitable for wall-bounded flows, a series of empirical functions are introduced to resolve the boundary layers with some near-wall mesh requirements attached, which have to be satisfied to reach a meaningful solution. Therefore, whenever the parameters such as blowing ratio change, the $y^+$ value will change and the near-wall mesh has to be changed accordingly.

It was shown that the methodology was very reliable and could yield accurate results over a wide range of blowing ratios and density ratios. When used appropriately, this method successfully captured the jet liftoff effect in traditional circular jet-in-cross flow over a flat plate. Figure 3 shows that the current methodology yields a prediction much closer to the experimental data than previous studies [20–22,26–28]. Figure 4 shows that the realizable $k$–$\epsilon$ model outperforms all the other turbulence models tested. Therefore, in the present work, the realizable $k$–$\epsilon$ model has been selected to perform the simulations by solving the Reynolds-averaged Navier–Stokes equations.

After convergence is reached, the adiabatic effectiveness $\eta$ can be calculated as

$$\eta = \frac{T_{\mathrm{aw}} - T_\infty}{T_j - T_\infty} \tag{5}$$

In this study, the same method as previous experimental work will be used to calculate the heat transfer coefficient. A first case was run until convergence, and the heat transfer coefficient with film cooling was determined from

$$h_f = \frac{q''}{T_w - T_{\mathrm{aw}}} \tag{6}$$

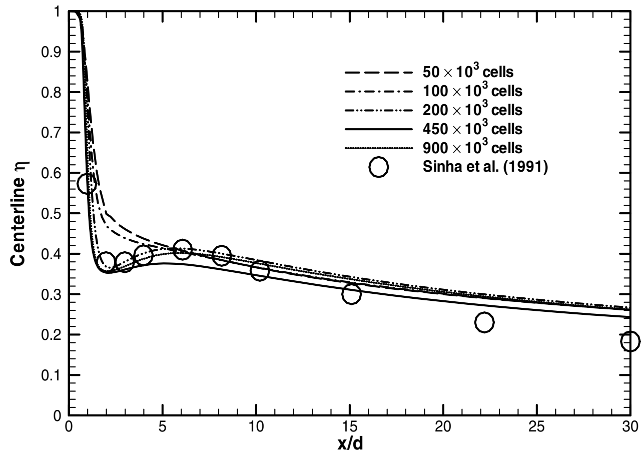
where $T_{\mathrm{aw}} = T_\infty$ because the temperature of coolant is forced to be equal to that of the mainstream. Because this study used a multiblock structured mesh, a second case was run with the jet and plenum sections removed, and the heat transfer coefficient without film cooling was determined from

$$h_o = \frac{q''}{T_w - T_\infty} \tag{7}$$

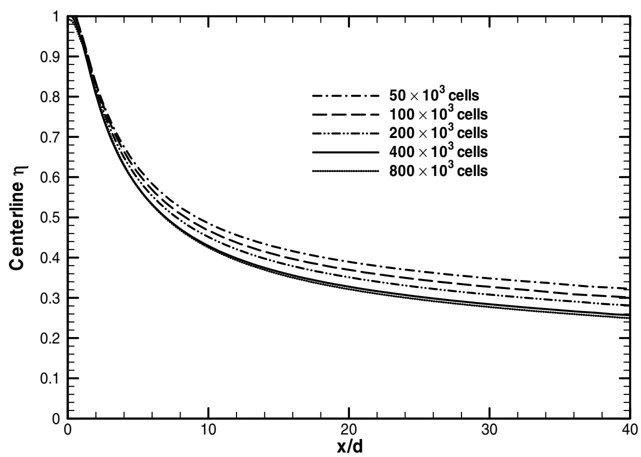
In either case, the constant heat flux boundary condition was applied on the bottom wall of the test section, downstream of injection, as in most experiments. It should be noted that the effect of impingement cooling on the blade material was not taken into account due to the fact that either adiabatic or constant heat flux boundary condition was applied on the test section only, downstream of the jet exits, for comparison purposes.

## IV. Results and Discussion

Figure 5 shows the grid independence for the cases of a single row of circular holes and the new scheme of two rows of staggered holes. To capture the fine features of the jet liftoff effect for a circular hole at high blowing ratio a mesh requires a minimum of $200 \times 10^3$ cells. Although more cells may lead to a better solution, the computing time increases exponentially. A mesh of between $300 \times 10^3$ and $600 \times 10^3$ cells is appropriate to reach an acceptable accuracy with a reasonable running time. For the new scheme, as jet liftoff does not occur at high blowing ratios, much fewer cells are needed to reach a result with reasonable accuracy. If jet liftoff is not

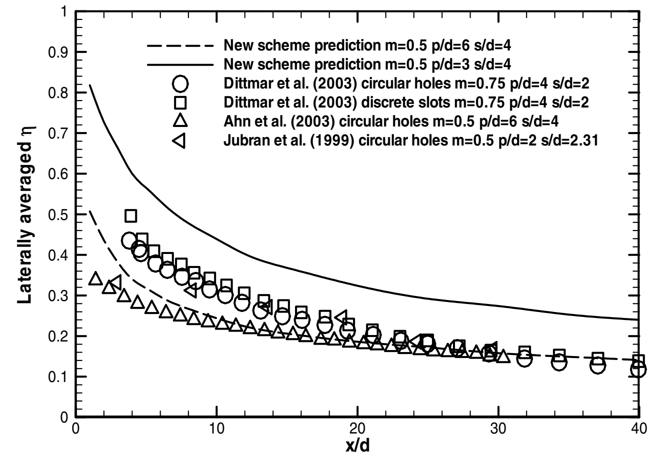



**a) Circular hole test m = 1, D.R. = 2**

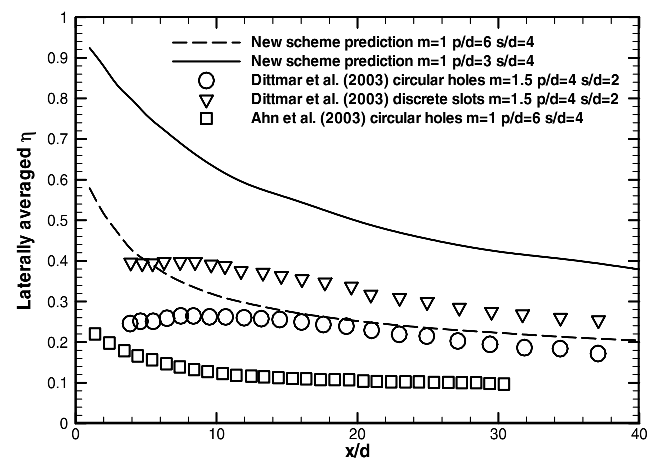



**b) New scheme two rows of holes m = 0.5, D.R. = 2**

**Fig. 5 Grid independence, a) circular hole and b) new scheme.**




**a) Two rows of staggered holes at low blowing ratio**




**b) Two rows of staggered holes at high blowing ratio**

**Fig. 6 Comparison of laterally averaged $\eta$ between the new scheme and other schemes.**

present with associated flow separation and recirculation in the flowfield, the same number of cells could produce results with higher accuracy.

Comparisons of laterally averaged cooling effectiveness for the new scheme with two rows of holes and other schemes are presented in Fig. 6. At low blowing ratios, the jets in all the different cooling schemes stay attached to the wall after injection. The prediction of the new scheme is slightly higher than or at least at the same level as the experimental data of other schemes [13,15,19], as shown in Fig. 6a. Reducing the pitch ratio results in a higher level of laterally averaged effectiveness because less protected surface area shares the same amount of coolant. At high blowing ratios, Fig. 6b shows that the new scheme consistently provides higher effectiveness than other schemes such as circular holes with compound angles (Ahn et al. [15]) and discrete slots (Dittmar et al. [19]). Comparison between the dashed line and the square symbols shows that at the same pitch and spacing ratio for the two rows of staggered holes, the new scheme offers an effectiveness twice that of circular holes with compound angles because the circular holes lift off from the surface at high blowing ratios in spite of compound angles. The advantages of the new scheme can also be seen in Fig. 7, in which comparisons with other shaped holes [4–6,28] were made for a single row of holes. These comparisons were made due to the lack of experimental data of shaped holes in more than one row arrangements.

As the blowing ratio increases, the coolant momentum also increases, which causes greater interaction between the jets and the mainstream. In the circular holes with a certain angle, usually 30–60 deg to the streamwise direction, the interaction between the jets and the mainstream creates a counter-rotating secondary flow within the jet trajectory. At high blowing ratios, high jet velocity produces a low pressure region at the trailing edge immediately after

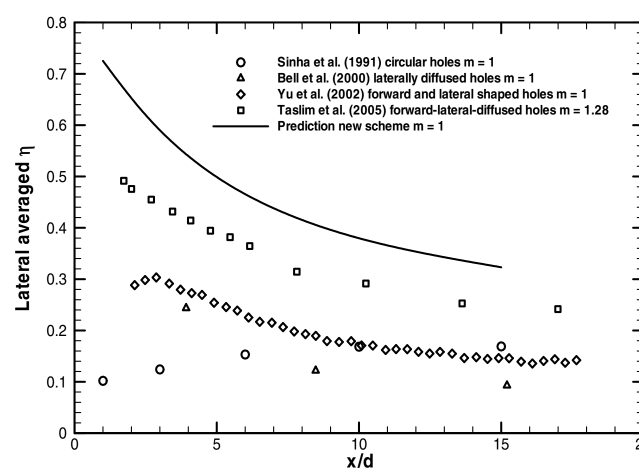



**Fig. 7 Laterally averaged $\eta$ for different schemes in one row of holes.**

injection, which sucks in some hot mainstream air with the aid of secondary vortices. Ensuing flow separation and recirculation cause a serious deterioration of protection. A high pressure region at the leading edge of the jet due to the impingement of the mainstream on the jet, coupled with the low pressure region at the trailing edges, causes the jet bend toward downstream. After a certain distance downstream of the exits, turbulence causes part of the coolant to be dissipated into the mainstream, therefore reducing its momentum. This reduction allows the mainstream to push the jet back to the wall and reattachment occurs. In the new scheme, a sudden reduction of the jet momentum occurs after the coolant enters the bend. This is due to a sudden increase in cross-sectional area. Additionally, the bend changes the momentum of the jet from upward to the streamwise direction. Further downstream towards the exit, the flow path area increases gradually and the coolant momentum was further reduced, therefore enlarging the coverage area. All these factors contribute to enhancing surface protection by eliminating flow separation and recirculation. With this new scheme, the jet liftoff phenomenon is avoided and the cooling effectiveness increases monotonically with blowing ratio.

Figure 8 shows comparisons of arrangements with two rows of holes. The staggered arrangement provides higher laterally averaged effectiveness than the inline arrangement, consistent with that of circular hole results from the literature. Because the coolant does not spread well in the spanwise direction for the inline arrangement, much of the coolant is quickly diffused into the mainstream, resulting in a lower laterally averaged effectiveness. On the other hand, the staggered arrangement covers the protected surface more uniformly. Consequently, the use of coolant is more efficient because the neighboring jets can easily coalesce at the midspan. Also shown in this figure is the behavior of one, two, and three rows of holes for the new scheme. It can be seen that two rows of holes provide higher effectiveness than one row of holes and three rows of holes yield higher effectiveness than two rows of holes. Nevertheless, the increase in effectiveness from two to three rows is less than half of the increase from one to two rows. This indicates that a further increase in the number of rows beyond two only gains limited increase in cooling effectiveness. Thus, it is of major importance to have the surface covered with coolant. However, surface coverage by additional coolant layers from more than two neighboring jets yields a less efficient use of the coolant.

Figure 9 shows the contours of adiabatic cooling effectiveness at the wall for the three configurations. Figure 9a demonstrates that the two inline rows do not provide good lateral coolant spreading with the high cooling effectiveness area concentrated mainly around the centerline, leaving the midspan unprotected. Shaping reduces the momentum of the jet significantly, resulting in very weak vortices, and the expanded exit stretches the coolant thin after the injection. The spanwise momentum component of the jets from the laterally expanding exits is too weak to spread coolant in the spanwise direction under the strong influence of the mainstream. At high blowing ratios, the mainstream is strong enough to break into the jet at the leading edge of the exit before the injection. The two staggered rows provide more even protection, as shown in Fig. 9b, simply because the jets are better positioned, causing more uniform distribution of the coolant after injection. The three

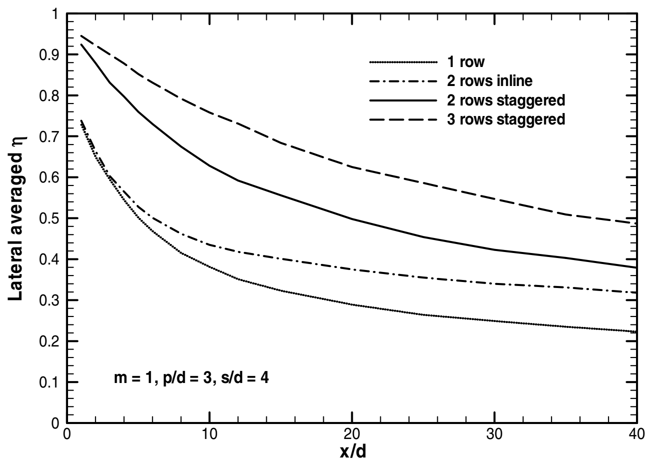
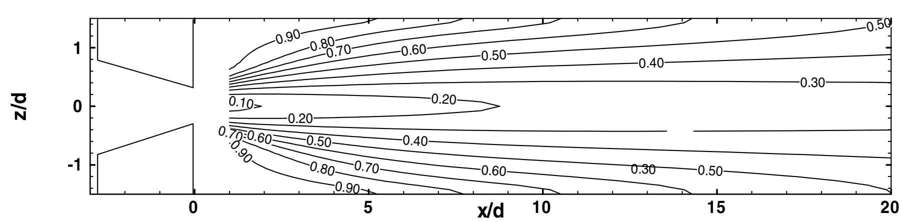
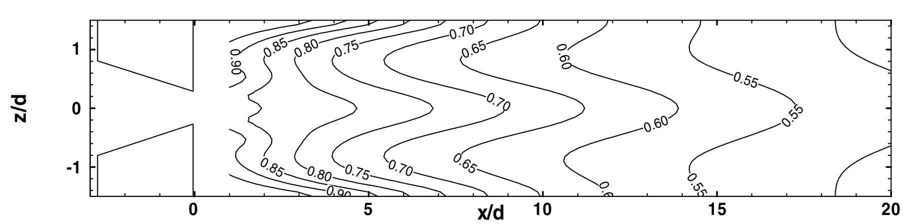


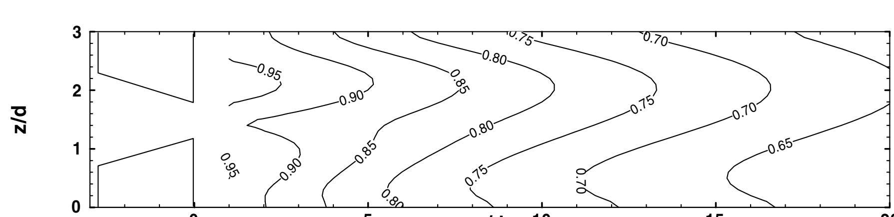

**Fig. 8 Laterally averaged $\eta$ for one, two, and three rows of the new scheme.**




**a) Two inline rows**




**b) Two staggered rows**




**c) Three staggered rows**

**Fig. 9 Contours of $\eta$ on the test surface.**

staggered rows provide slightly higher cooling effectiveness, compared with two rows, but the coverage is no more uniform, as shown in Fig. 9c.

Figure 10 shows the normalized streamwise velocity profiles on the central plane at a blowing ratio of 1. The thickness of boundary layer is increasing downstream after injection, as can be seen from the comparison of velocity profiles at different locations. The velocity gradually increases further away from the wall, signifying that the coolant stays close to the wall without penetration. At $x/d = 1$, the injection of the jets causes a large disturbance to the boundary layer, particularly for the two inline rows arrangement, as shown in Fig. 10a. This disturbance is relaxed quickly after $x/d = 4$. The thickness of the coolant layer at the central plane for two staggered rows is substantially thinner than that of two inline rows at all locations after injection. Therefore, coolant accumulates at the centerline of the two inline rows arrangement after the injection of the second jet, resulting in poor spreading. Although the three rows arrangement has roughly the same coolant layer thickness as the two

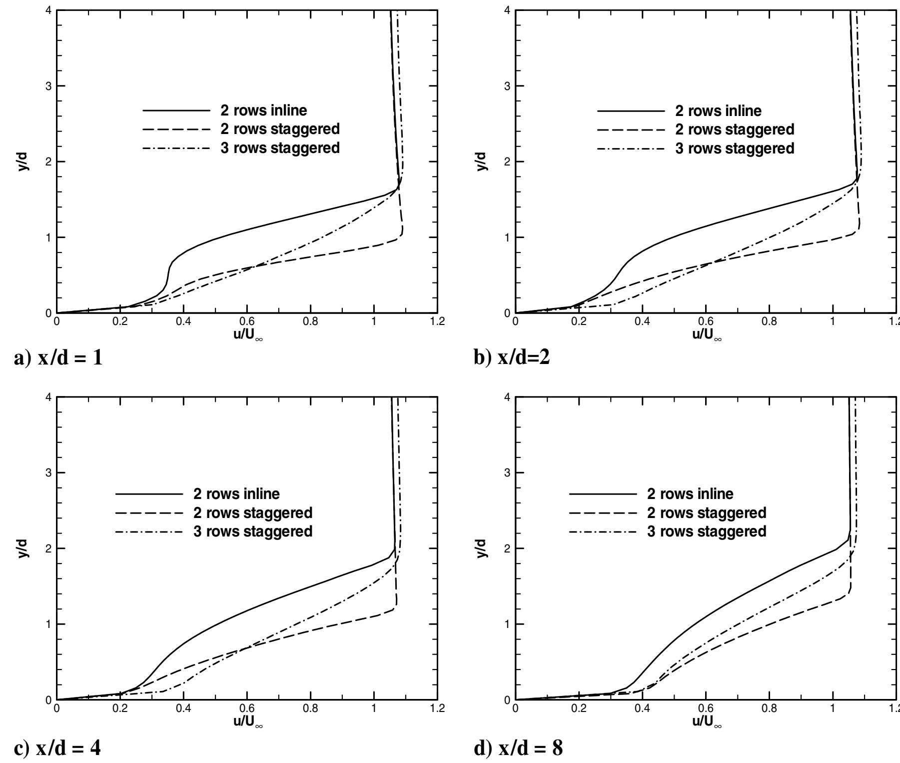



**Fig. 10 Profiles of the streamwise velocity on the central plane for $m = 1$.**

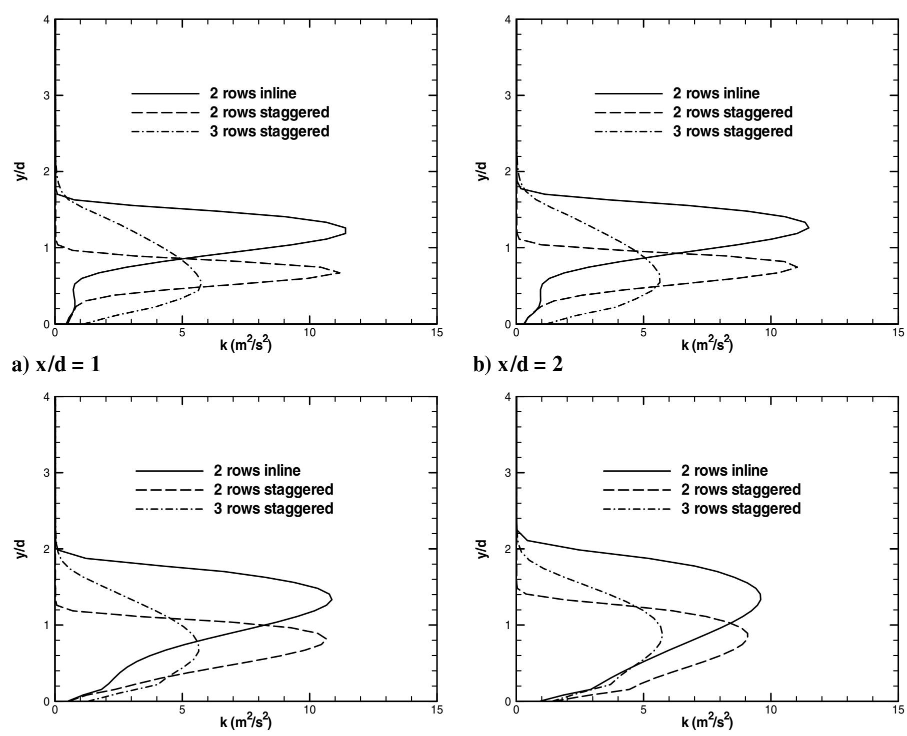
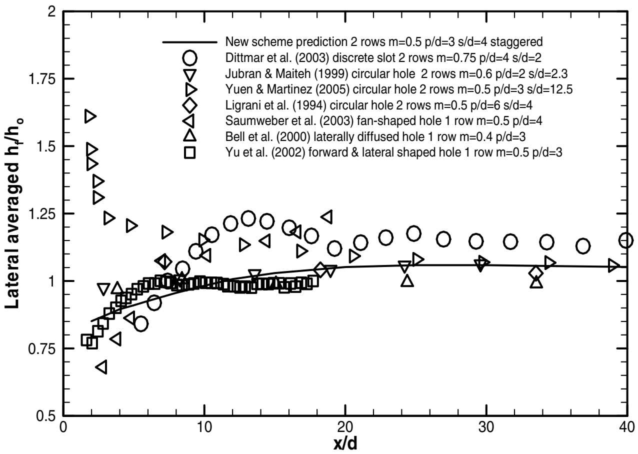
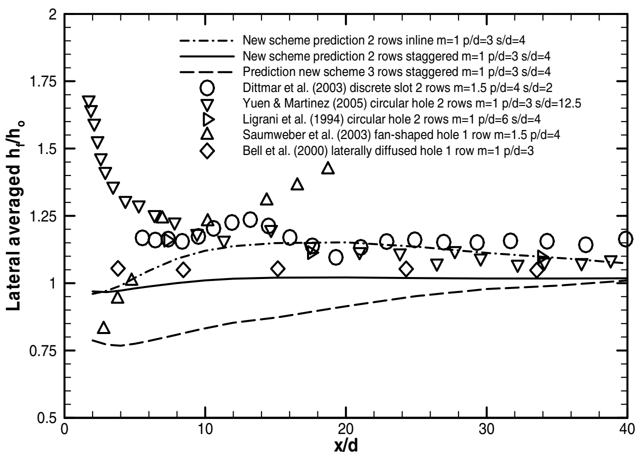



**Fig. 11 Profiles of the $k$ on the central plane for $m = 1$.**




a) At low blowing ratio




b) At high blowing ratio

**Fig. 12 Comparison of laterally averaged $h_f/h_o$ between the new scheme and other schemes.**

inline rows, this is only because it has 50% more coolant to distribute on the wall.

Figure 11 shows the turbulence kinetic energy profiles along the centerline at a blowing ratio of 1. Among these three configurations, the two inline rows generate the highest level of turbulence, whereas three staggered rows have the lowest turbulence level. The accumulation of coolant on the wall after injection pushes the maximum level of turbulence further from the wall with two inline rows as compared with two or three staggered rows. It is also observed that the area of high turbulence in the two inline rows arrangement is larger than the two staggered rows counterpart. This causes greater coolant mixing with the mainstream. Consequently, there is a lower level of laterally averaged effectiveness with the inline arrangement. The three staggered rows create a larger area of turbulence in the vertical direction than the two row configurations. However, its turbulence level is considerably lower with subsequent lower mixing and higher effectiveness.

Figure 12 shows the heat transfer coefficient of the new scheme in comparison with other cooling schemes published in recent years . It was determined that at a low blowing ratio (Fig. 12a), the new scheme yields nearly the same level of heat transfer as other schemes [4,6,13,14,17,19,29]. At a high blowing ratio (Fig. 12b), a lower level of heat transfer compared with previous cooling schemes [4,14,17,19,29] is predicted because the new scheme prevents jets from penetrating into the mainstream, resulting in less mixing with the mainstream and better coverage. Figure 12b also demonstrates the prediction of two inline rows and three staggered rows. The inline arrangement provides higher heat transfer levels because coolant builds up along the centerline after injection of the second jet, as compared with the staggered arrangement, giving rise to poor coverage of the surface. Three rows of holes offer a lower level of laterally averaged heat transfer than two rows. Interesting here is that three rows provide a heat transfer level considerably lower than two rows of holes whereas the heat transfer level of two rows of holes lingers around one. This is caused by additional coolant being injected by the three rows arrangement. Hence, a thicker coolant boundary layer reduces heat transferred to the protected surface. Comparison of heat transfer levels for the new scheme between high and low blowing ratios shows that the heat transfer value is insensitive to the blowing ratios, lingering around one. This is in contrast to the circular holes scheme in which the heat transfer level increases with blowing ratio because high blowing ratio causes higher levels of mixing and penetration. Therefore, high blowing ratios can be used with the new scheme to generate more protection without concern about raising heat transfer levels.

## V. Conclusions

In the present study, a methodology with high fidelity developed and validated in a number of benchmark cases in previous studies has been applied to a new scheme with two and three rows of holes. To make the results of predictions of the new scheme comparable to those of the other schemes, from the experimental works in the literature, the dimension of the new scheme was scaled appropriately. Because of a lack of available experimental data for multiple rows of holes, either shaped or circular, some comparisons were made between the predictions of the new scheme and experimental data of other schemes under different parameters, such as pitch ratio, blowing ratio, and row spacing. After all the differences were considered, the following conclusions emerged from this study:

1) The proposed cooling scheme can prevent jets from lifting off the surface at high blowing ratios and provides the highest cooling effectiveness among the schemes compared. The new cooling scheme also produces a considerably lower heat transfer coefficient in the near-hole region, as compared with circular hole. This is particularly true at high blowing ratios when circular holes undergo liftoff.

2) The two staggered rows of holes provide much better surface protection than two inline rows. The staggered arrangement offers much higher laterally averaged cooling effectiveness with a slightly lower heat transfer coefficient. In addition, in the staggered arrangement, both the cooling effectiveness and heat transfer results are more uniform.

3) Three staggered rows give slightly higher cooling effectiveness than its two rows counterpart but with considerably lower heat transfer results. From the viewpoint of surface cooling, two staggered rows are highly recommended when row spacing is small, because three rows of holes offer only a 10% increase in cooling effectiveness with a 50% coolant increase.

4) Reducing the pitch ratio in half from 6 to 3 roughly doubles the laterally averaged cooling effectiveness partly because the jet liftoff effect at high blowing ratios is avoided leading to more uniform coverage of the protected surfaces and partly because jets 6*d* apart are too far from each other to coalesce easily after injection, leaving the midspan unprotected.

5) The heat transfer results for the two rows of holes, whether inline or staggered or at different pitch ratios, are insensitive to blowing ratios with the normalized averaged heat transfer coefficient lingering around one. This is a desirable characteristic because high blowing ratios can be used to generate a higher cooling effectiveness.

The experimental study of the new cooling scheme is in progress by the same group at Concordia University and the validation of CFD code can be done once experimental data is made available. In addition, a conjugate analysis of the new scheme considering the effect of impingement cooling and the performance of the new scheme on an actual aerofoil are prospective topics to be studied in the future.

## Acknowledgments

The authors are grateful for the support provided by Pratt and Whitney Canada. The computational facilities in the CFD laboratory at Concordia University making this work possible are also gratefully acknowledged.

## References

[1] Bunker, R. S., "A Review of Shaped Hole Turbine Film-Cooling Technology," *Journal of Heat Transfer*, Vol. 127, No. 4, 2005, pp. 441–453.

[2] Schmidt, D. L., Sen, B., and Bogard, D. G., "Film Cooling with Compound Angle Holes: Adiabatic Effectiveness," *Journal of Turbomachinery*, Vol. 118, Oct. 1996, pp. 807–813.

[3] Thole, K., Gritsch, M., Schulz, A., and Wittig, S., "Flowfield Measurements for Film-Cooling Holes with Expanded Exits," *Journal of Turbomachinery*, Vol. 120, April 1998, pp. 327–336.

[4] Bell, C. M., Hamakawa, H., and Ligrani, P. M., "Film Cooling from Shaped Holes," *Journal of Heat Transfer*, Vol. 122, May 2000, pp. 224–232.

[5] Taslim, M. E., and Khanicheh, A., "Film Effectiveness Downstream of a Row of Compound Angle Film Holes," *Journal of Heat Transfer*, Vol. 127, April 2005, pp. 434–440.

[6] Yu, Y., Yen, C. H., Shih, T. I. P., and Chyu, M. K., "Film Cooling Effectiveness and Heat Transfer Coefficient Distributions Around Diffusion Shaped Holes," *Journal of Heat Transfer*, Vol. 124, No. 5, 2002, pp. 820–827.

[7] Sen, B., Schmidt, D. L., and Bogard, D. G., "Film Cooling with Compound Angle Holes: Heat Transfer," *Journal of Turbomachinery*, Vol. 118, Oct. 1996, pp. 800–806.

[8] Lee, H. W., Park, J. J., and Lee, J. K., "Flow Visualization and Film Cooling Effectiveness Measurements Around Shaped Holes with Compound Angle Orientations," *International Journal of Heat and Mass Transfer*, Vol. 45, No. 1, 2002, pp. 145–156.

[9] Gritsch, M., Colban, W., Schar, H., and Dobbeling, K., "Effect of Hole Geometry on the Thermal Performance of Fan-Shaped Film Cooling Holes," *Journal of Turbomachinery*, Vol. 127, No. 4, 2005, pp. 718–725.

[10] Giebert, D., Gritsch, M., Schulz, A., and Wittig, S., "Film-Cooling from Holes with Expanded Exits: A Comparison of Computational Results with Experiments," American Society of Mechanical Engineers Paper 97-GT-163, 1997.

[11] Hyams, D. G., and Leylek, J. H., "A Detailed Analysis of Film-Cooling Physics: Part 3–Streamwise Injection with Shaped Holes," *Journal of Turbomachinery*, Vol. 122, No. 1, 2000, pp. 122–132.

[12] Bohn, D., Ren, J., and Kusterer, K., "Conjugate Heat Transfer Analysis of Film Cooling Configurations with Different Hole Geometries," American Society of Mechanical Engineers Paper GT2003-38359, 2003.

[13] Jubran, B. A., and Maiteh, B. Y., "Film Cooling and Heat Transfer from a Combination of Two of Simple and/or Compound Angle Holes in Inline and/or Staggered Configuration," *International Journal of Heat and Mass Transfer*, Vol. 34, No. 6, 1999, pp. 495–502.

[14] Ligrani, P. M., Wigle, J. M., Ciriello, S., and Jackson, S. M., "Film-Cooling from Holes with Compound Angle Orientations: Part 1—Results Downstream of Two Staggered Rows of Holes with 3d Spanwise Spacing," *Journal of Heat Transfer*, Vol. 116, May 1994, pp. 341–352.

[15] Ahn, J., Jung, I. S., and Lee, J. K., "Film Cooling from Two Rows of Holes with Opposite Orientation Angles: Injection Behavior and Adiabatic Film Cooling Effectiveness," *International Journal of Heat and Fluid Flow*, Vol. 24, No. 1, 2003, pp. 91–99.

[16] Yuen, C. H. N., and Martinez-Botas, R. F., "Film Cooling Characteristics of Rows of Round Holes at Various Streamwise Angles in a Crossflow: Part 1. Effectiveness," *International Journal of Heat and Mass Transfer*, Vol. 48, No. 23–24, 2005, pp. 4995–5016.

[17] Yuen, C. H. N., and Martinez-Botas, R. F., "Film Cooling Characteristics of Rows of Round Holes at Various Streamwise Angles in a Crossflow: Part 2. Heat Transfer Coefficients," *International Journal of Heat and Mass Transfer*, Vol. 48, No. 23–24, 2005, pp. 5017–5035.

[18] Maiteh, B. Y., and Jubran, B. A., "Effects of Pressure Gradient on Film Cooling Effectiveness from Two Rows of Simple and Compound Angle Holes in Combination," *Energy Conversion and Management*, Vol. 45, No. 9–10, 2004, pp. 1457–1469.

[19] Dittmar, J., Schulz, A., and Wittig, S., "Assessment of Various Film-Cooling Configurations Including Shaped and Compound Angle Holes Based on Large-Scale Experiments," *Journal of Turbomachinery*, Vol. 125, No. 1, 2003, pp. 57–64.

[20] Leylek, J. H., and Zerkle, R. D., "Discrete-Jet Film Cooling: A Comparison of Computational Results with Experiments," American Society of Mechanical Engineers Paper 93-GT-207, 1993.

[21] Walters, D. K., and Leylek, J. H., "A Detailed Analysis of Film Cooling Physics: Part 1: Streamwise Injection with Cylindrical Holes," *Journal of Turbomachinery*, Vol. 122, No. 1, 2000, pp. 102–112.

[22] Immarigeon, A. A., "Advanced Impingement/Film-Cooling Schemes for High-Temperature Gas Turbine—Numerical Study," M.Sc. Thesis, Concordia Univ., Montréal, 2004.

[23] Shih, T.-H., Liou, W. W., Shabbir, A., Yang, Z., and Zhu, J., "A New $k$-$\epsilon$ Eddy-Viscosity Model for High Reynolds Number Turbulent Flows—Model Development and Validation," *Computers and Fluids*, Vol. 24, No. 3, 1995, pp. 227–238.

[24] Zhang, X. Z., and Hassan, I., "An Advanced-Louver Cooling Scheme for Gas Turbines—Part 1 Effectiveness," American Society of Mechanical Engineers Paper HT2005-72074, 2005.

[25] Zhang, X. Z., and Hassan, I., "An Advanced-Louver Cooling Scheme for Gas Turbines—Part 2 Heat Transfer Performance," American Society of Mechanical Engineers Paper HT2005-72075, 2005.

[26] Ferguson, J. D., Walters, D. K., and Leylek, J. H., "Performance of Turbulence Models and Near-Wall Treatments in Discrete Jet Film Cooling Simulations," American Society of Mechanical Engineers Paper 98-GT-438, 1998.

[27] Mulugeta, K. B., and Patankar, S. V., "A Numerical Study of Discrete-Hole Film Cooling," American Society of Mechanical Engineers Paper 96-WA/HT-8, 1996.

[28] Sinha, A. K., Bogard, D. G., and Crawford, M. E., "Film-Cooling Effectiveness Downstream of a Single Row of Holes with Variable Density Ratio," *Journal of Turbomachinery*, Vol. 113, No. 3, 1991, pp. 442–449.

[29] Saumweber, C., Schulz, A., and Wittig, S., "Free-Stream Turbulence Effects on Film Cooling with Shaped Holes," *Journal of Turbomachinery*, Vol. 125, No. 1, 2003, pp. 65–72.