Numerical investigation of heat transfer on film cooling with shaped holes

X.-Z. Zhang and I. Hassan
Department of Mechanical and Industrial Engineering, Concordia University, Montréal, Québec, Canada

Abstract

Purpose – To develop a reliable methodology and procedure of simulating the jet-in-crossflow using the current turbulence models and numerically investigate the cooling performance of a new scheme for the engines of next generation.

Design/methodology/approach – A new advanced film cooling scheme is proposed based on the literature survey and a systematic methodology developed to successfully predict the right level of heat transfer in the CFD simulation of film cooling.

Findings – The proposed cooling scheme gives considerable lower heat transfer coefficient at the centerline in the near hole region than the traditional cylindrical hole, especially at a high blowing ratio when traditional cylindrical hole undergoes liftoff.

Research limitations/implications – The number of cooling holes in the computational domain is limited by the speed of the computers used.

Practical implications – The new methodology can be used to numerically test new cooling schemes in the design of turbine blades and to provide useful information/data under actual working conditions to design engineers.

Originality/value – This paper provides some useful information on the simulation of film cooling in terms of the performance of different turbulence models and wall treatments and also sends some valuable messages regarding the design of cooling scheme of turbine blades to the technical community.

Keywords Films (states of matter), Cooling, Heat transfer

Paper type Research paper

Nomenclature

d = hydraulic diameter of hole at exit plane (m)
h = heat transfer coefficient (W/m^{2}K), $h = q''/(T_w - T_{aw})$
h_o = heat transfer coefficient without film cooling (W/m^{2}K), $h_o = q''/(T_w - T_\infty)$
k = turbulent kinetic energy (m^2/s^2)
m = blowing ratio (or rate) $m = (\rho_j U_j)/(\rho_\infty U_\infty)$
Re_D = Reynolds number based on freestream velocity and injection diameter, $Re_D = (\rho_\infty U_\infty d)/\mu_\infty$
q'' = surface heat flux per unit area (W/m^2)
T = temperature (K)
U = velocity (m/s)
x = stream-wise coordinate (m)
y = vertical coordinate (m)
z = span-wise coordinate (m)
y^+ = non-dimensional wall distance ($y^+ = (\rho u_\tau y_p)/\mu$)
y_p = coordinate normal to the wall surface (m)

Greek symbols

ε = dissipation rate of turbulent kinetic energy (m^2/s^3)
η = local adiabatic film cooling effectiveness ($\eta = (T_{aw} - T_\infty)/(T_j - T_\infty)$)
ρ = density (kg/m^3)

International Journal of Numerical Methods for Heat & Fluid Flow
Vol. 16 No. 8, 2006
pp. 848-869

0961-5539
DOI 10.1108/09615530610702032

Subscripts and superscripts
aw = adiabatic wall
f = conditions with film cooling
o = conditions in the absence of film cooling
j = refers to the jet
w = wall conditions
∞ = mainstream conditions at inlet plane and in freestream

1. Introduction

In order to increase their efficiency, the inlet temperatures of gas turbines have been raised significantly in the last three decades. Modern gas turbine engines typically operate at inlet temperatures of 1,800-2,000 K, which is far beyond the allowable metal temperature. Film cooling becomes increasingly important in the aerospace industry in cooling of the turbine blades. Many experimental and computational studies have been conducted in order to study the cooling process of gas turbine blades, understand this complex flow and heat process, and devise the best possible cooling schemes.

Methods that have been mainly investigated include heat transfer and film cooling (Hyams and Leylek, 2000; Cho *et al.*, 2001; Gartshore *et al.*, 2001; Goldstein and Jin, 2001; Cutbirth and Bogard, 2002; Yuen and Martinez-Botas, 2003), impingement cooling (Son *et al.*, 2001; Taslim *et al.*, 2001; Li *et al.*, 2001), and advanced internal or external cooling (Azad *et al.*, 2000; Taslim *et al.*, 2001). These methods are commonly studied with the following parameters: injection orientations (Brittingham and Leylek, 2000; Jung and Lee, 2000; Gritsch *et al.*, 2001; Dittmar *et al.*, 2003), hole length (Burd *et al.*, 1998; Harrington *et al.*, 2001), free stream turbulence (Ekkad *et al.*, 1998; Al-Hamadi *et al.*, 1998; Maiteh and Jubran, 1999; Mayhew *et al.*, 2003; Saumweber *et al.*, 2003), hole entrance effects (Hale, 1999; Wilfert and Wolff, 2000; Brittingham and Leylek, 2000; Gritsch *et al.*, 2003), hole exit tapering (Kohli and Bogard, 1999; Sargison *et al.*, 2002; Nasir *et al.*, 2003), hole exit expanding (Yu *et al.*, 2002; Rhee *et al.*, 2002; Sargison *et al.*, 2002; Gritsch *et al.*, 2003; York and Leylek, 2003; Kim and Kim, 2004), surface roughness (Schmidt *et al.*, 1996) and density ratio effects (Ammari *et al.*, 1990; Ekkad *et al.*, 1998), and measurement techniques (Sen, 1996; Goldstein and Jin, 2001; Ai *et al.*, 2001; Baldauf *et al.*, 2002; Vogel *et al.*, 2003; Cho *et al.*, 2001; Yuen and Martinez-Botas, 2003). Only recent references have been given here and earlier reports may be traced through the reference lists of the papers cited. From these reports and their references, some broad generalizations can be made about the methods, geometries, and conditions affecting the heat transfer performance of internal or external cooling schemes of gas turbine blades:

- At both high and low blowing ratio, for traditional cylindrical hole, the heat transfer asymptotically approaches $h_f/h_o = 1$ for $x/d > 10$. The magnitude of h_f/h_o increases with increasing blowing ratio. Higher blowing ratio causes high penetration and mixing with the mainstream, resulting in an enhanced heat transfer coefficient and reduced lateral jet spread. The heat transfer coefficient decreases rapidly for the first ten diameters downstream of the hole, then gradually decreases farther downstream.
- Holes with expanded exits have lower heat transfer coefficients at elevated blowing ratios when compared to cylindrical holes. Variation of the relative spacing or the free-stream turbulence level resulted only in slight changes of the heat transfer coefficient. Increasing the mainstream turbulence intensity results in an increase in the local heat transfer level, especially at low blowing ratio.

- The magnitude of the heat transfer coefficient along the centerline is the lowest for any given hole geometry and the greatest around the side-edge of an injection hole due mainly to the interaction between the injection and the mainstream. Normal jets penetrate deeper and interact more with the mainstream, resulting in higher heat transfer coefficients. With inclined jets, the interaction with the mainstream is lower, and shows less of an increase in heat transfer.
- In contrast to the fact that all the results of adiabatic effectiveness from different researchers are similar to one another, the results of the heat transfer coefficient by different researchers vary widely depending on how the constant heat flux is instrumented. Conduction error in the span-wise direction on the test surface is much larger than believed and would tend to smooth out the heat transfer coefficient from the true distribution when stainless steel heater foils are employed in construction of constant heat flux plate.
- Geometry and mesh system exert considerable influence on the numerical solutions. The effect of the geometry and the mesh on the solution could be so large that the performance of the turbulence models and wall treatments can be completely masked and their performance cannot be accurately evaluated. It is necessary to include the plenum and film hole in the computational domain in order to ensure a realistic profile at the exit of the jet, and to account for the interaction between the mainstream, the jet, and the plenum.
- The numerical simulation is not very reliable and predictions vary significantly with the different turbulence models. The incapacity of the computational model to predict the surface heat transfer coefficient, with an over-predicted error between 40 and 100 percent for the span-wise-averaged heat transfer coefficient and associated local errors up to 200 percent for local heat transfer coefficient, was attributed to deficiencies in the near-wall treatment and turbulence modeling.

The present study will show that the conduction errors on the test surface are much larger than believed in previous experimental investigations. In order to measure the heat transfer coefficient, a constant heat flux condition on the bottom wall is required and the manner in which the constant heat flux plate is constructed has a significant effect on the temperature distribution on the bottom surface. This explains why there are large discrepancies between experimental results given by different researchers, as shown in Figure 1. Different definitions of heat transfer coefficients and experimental uncertainties could not satisfactorily explain the opposite trends. In this paper, the heat conduction on the test surface will be considered in the simulations by modeling conjugate heat transfer.

The heat transfer performance of a new cooling scheme will be presented, and 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 in terms of the heat transfer coefficient. With the concept of the proposed advanced cooling scheme – first designed by Immarigeon (2004) in collaboration with Pratt and Whitney Canada and refined by the authors to show its full advantages – a greater portion of the airfoil is protected. The film hole is designed in such a way that the coolant must go through a bend before exiting the blade, as shown in Figure 2, thus impinging on the blade material. Finally, the flow exits very close to the blade surface, minimizing

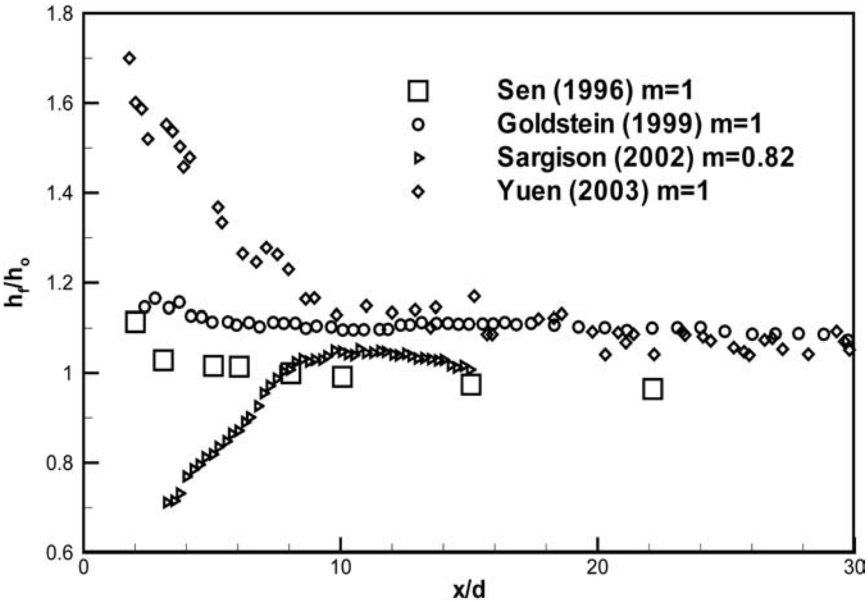

Figure 1.
Disparities between experimental data of *h* given by different researchers on traditional cylindrical hole

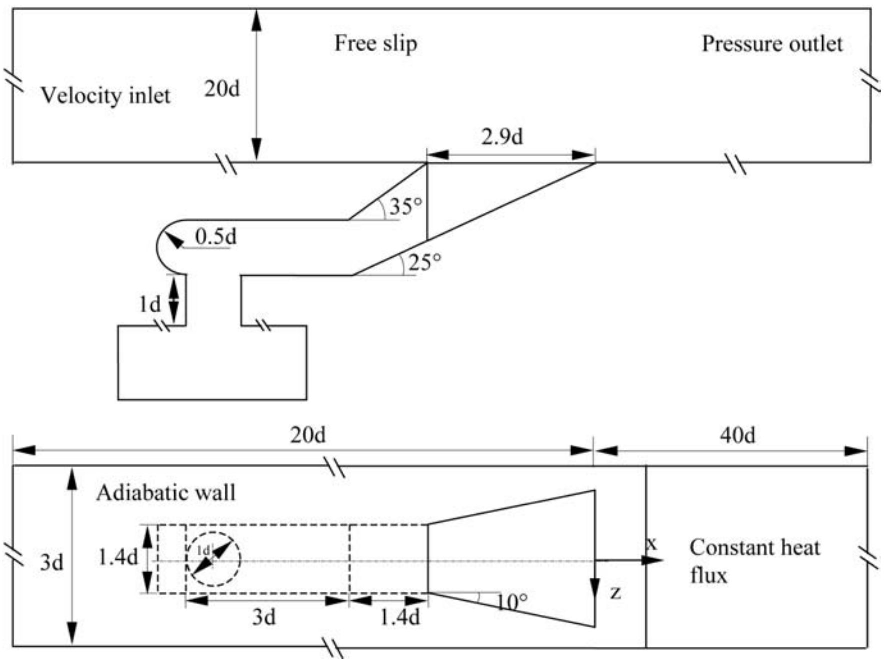

Figure 2.
Computational domain and boundary conditions for the new scheme

aerodynamic losses. This study systematically investigated the capability of the current turbulence models to predict the right level of heat transfer coefficient in different schemes, i.e. long circular jets without plenum, short circular jets with plenum, shaped holes and a new scheme.

2. Turbulence modeling and boundary conditions

The computational domains and film cooling geometries of the new scheme are shown in Figure 2. Each domain consists of an infinite row of film cooling holes in a flat plate, such that the end-wall effects are neglected. The parameters and the geometries of all the other different cases in the present computational study are exactly the same as in their corresponding experimental counterparts by Eriksen (1971), Sen (1996), Bell *et al.* (2000), and Yu *et al.* (2002). At the upstream inlet, a velocity inlet condition is applied, and at the outlet, a pressure outlet boundary condition is applied. The domain extends 20d from the test surface, which is far enough such that a free slip boundary condition or zero shear stress may be applied. If the computational domain is symmetric about the central plane, symmetry boundary conditions are imposed at both the central plane and at the mid-span plane. Otherwise, periodic boundary condition will be applied. At the bottom wall, from the point of upstream inlet to the starting point of the constant heat flux, an adiabatic wall boundary condition with no-slip is applied. From the starting point of the heat flux to outlet of the duct, a constant heat flux is imposed. On the other walls, an adiabatic wall boundary condition with no-slip is imposed.

In this study, four classes of different turbulence models provided by commercial software FLUENT are selected to perform the simulations by solving the Reynolds-averaged Navier-Stokes equations. The predictions using these models were validated and compared with the experimental data obtained by Eriksen (1971), Sen (1996), Bell *et al.* (2000), and Yu *et al.* (2002). As in the experiments, it is assumed that the flow is incompressible, steady-state turbulent flow with low turbulence intensity (<2 percent). Different meshes were used at the beginning to determine the optimum grid size and to ensure grid independent solution. The grid contained between 0.4 and 0.8 million cells when the standard wall functions were employed. When the enhanced wall treatment was selected, a grid independent solution was attained with a grid containing around one million cells.

Unlike the unstructured mesh, with usually tetrahedron elements, the structured mesh, with hexahedron as shown in Figure 3, cannot be adapted. Therefore, if the solution for y^+ is found to be out of the acceptable range, the mesh has to be discarded and a new one created. The typical procedure to reach a good solution using a structured mesh is outlined in Figure 4. Since y^+ is a solution dependent variable and not known a priori, it is pretty clear that it is time consuming to obtain a meaningful solution that meets the near wall mesh requirement. Physically, the boundary layer is comparably very thin; however, it has tremendous influence on the final solution. Hence, it is critically important to precisely control the near wall mesh y^+ according to the near wall mesh requirement. From this point of view, whenever the parameters such as blowing ratio change, the y^+ will change and the near wall mesh has to be changed in order to reach a meaningful solution. From this point of view, every mesh corresponds to a unique flow condition. In this study, the value of y^+ falls in the range of $30 \sim 60$ when standard wall functions are employed and on the order of one when two layer model is selected, dictated by the commercial software FLUENT.

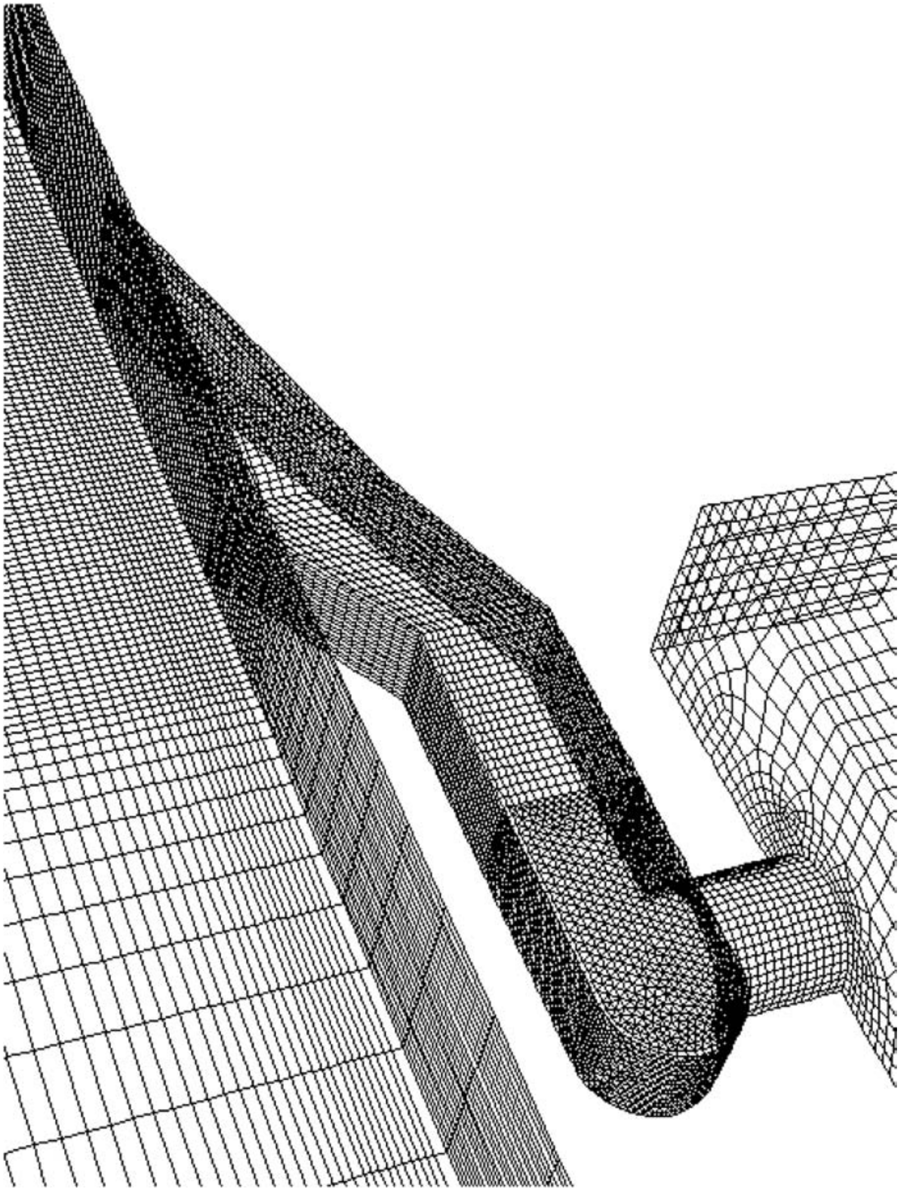

Figure 3.
Multi-block structured
mesh for the new scheme

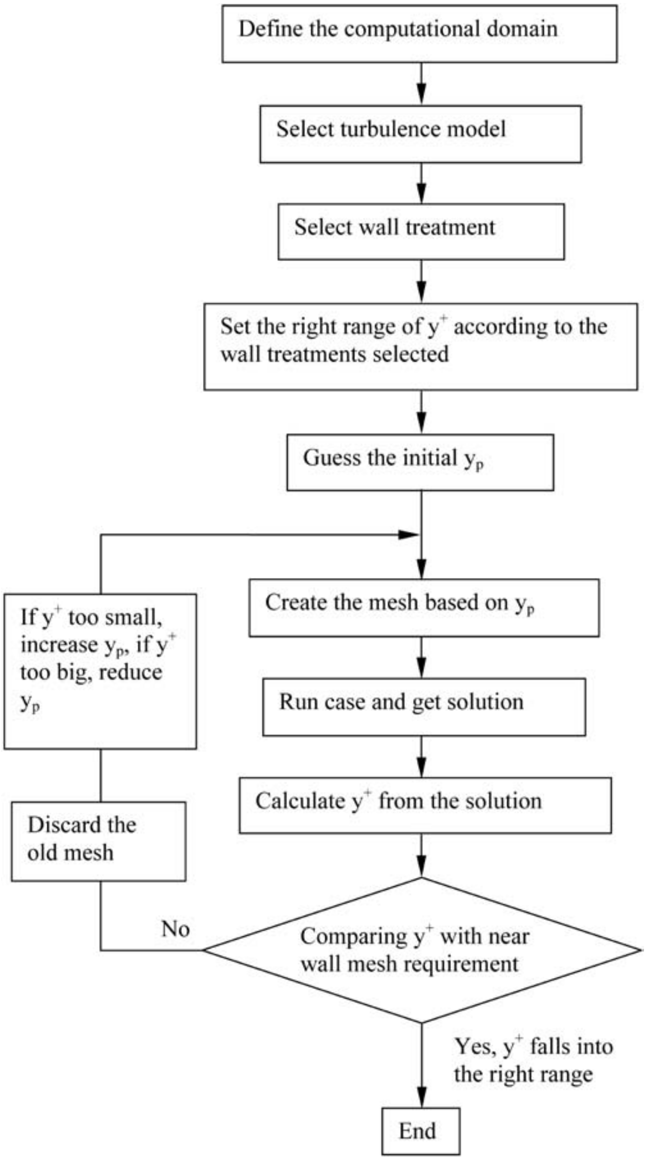

Figure 4.
Typical procedure to reach a good mesh and a meaningful solution

3. The calculation of the heat transfer coefficients

Unlike the adiabatic effectiveness η, which can be calculated as:

$$\eta = \frac{T_{\mathrm{aw}} - T_\infty}{T_{\mathrm{j}} - T_\infty}, \tag{1}$$

once T_{aw} becomes available from the solution at different mainstream and coolant temperatures, after the Navier-Stokes equations are solved numerically, the heat transfer coefficient is generally non-dimensionalized as $h_{\mathrm{f}}/h_{\mathrm{o}}$ in the literature. Non-dimensionalizing parameters is a good practice in most circumstances, however, the real trend of heat transfer could be significantly masked in this particular situation due to the fact that both h_{f}, the heat transfer coefficient with injection of coolant, and h_{o}, the heat transfer coefficient without injection of coolant, are a function of space, changing from position to position in the stream-wise direction. Probably, that is why some researchers presented the centerline heat transfer coefficient in the literature. In this study, the same way as in the experimental work will be employed in the calculation of the heat transfer coefficient. The first case was run until convergence and the heat transfer coefficient with film cooling is calculated as:

$$h_{\mathrm{f}} = \frac{q''}{T_{\mathrm{w}} - T_{\mathrm{aw}}}, \tag{2}$$

where $T_{\mathrm{aw}} = T_{\infty}$ because the temperature of coolant is forced to be equal to that of the mainstream as in the experiment. As this study used a multi-block structured mesh, a second case was run with the jet and plenum sections deleted and the heat transfer coefficient without film cooling is calculated from the numerical solution as:

$$h_{\mathrm{o}} = \frac{q''}{T_{\mathrm{w}} - T_{\infty}}. \tag{3}$$

In either case, the constant heat flux boundary condition is applied on the test section of the bottom wall, downstream of injection, as in the experiments. Because in both cases, the meshes of the duct section are exactly the same, it is easy to normalize the heat transfer results.

4. Results and discussion

In the present study, in order to validate the methodology which will be used for the new scheme, as well as validate the selected turbulence models, the experimental works of Eriksen (1971), Sen (1996), Bell *et al.* (2000), and Yu *et al.* (2002) have been chosen as the benchmark cases. Eriksen *et al.*'s case consisted of a very simple long circular jet without plenum, where the flow at the exit of jet was fully developed since long tubes (around 1 m) were used, thus eliminating uncertainties introduced by jet exit profiles. In Sen *et al.*'s case, a short circular jet with plenum was used, which represents real engine applications more closely. Both Bell and Yu's cases were shaped holes, similar to the new cooling scheme.

Figure 5 shows the performance of different turbulence models in terms of the normalized heat transfer coefficient on the centerline at $m = 1$ and $Re_{\mathrm{D}} = 0.22 \times 10^5$ for Eriksen's (1971) case. The Reynolds-stress model gives a very good prediction in the region of $x/d < 14$ beyond which it under predicts the heat transfer coefficient significantly. Both the k-ω model and the Spalart-Allmaras model slightly under predict the heat transfer coefficient in the near hole region of $x/d < 15$ and slightly over predict the heat transfer coefficient in the region of $x/d > 20$. The k-ε model slightly under predicts the heat transfer coefficient in the near hole region of $x/d < 15$ as well, however, beyond this region it gives an excellent result. Figure 6 shows the performance of three

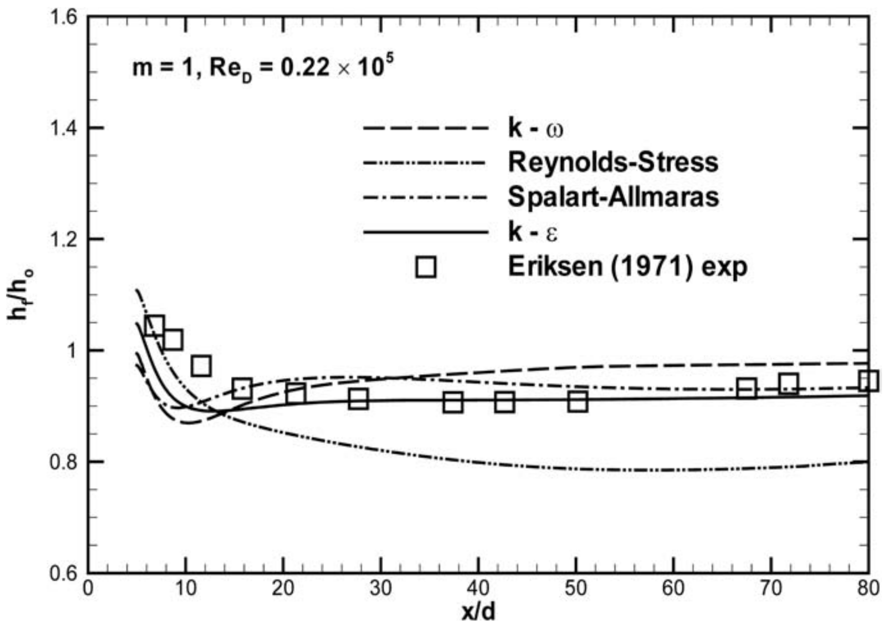

Figure 5.
The performance of turbulence models with the experimental data of Eriksen (1971)

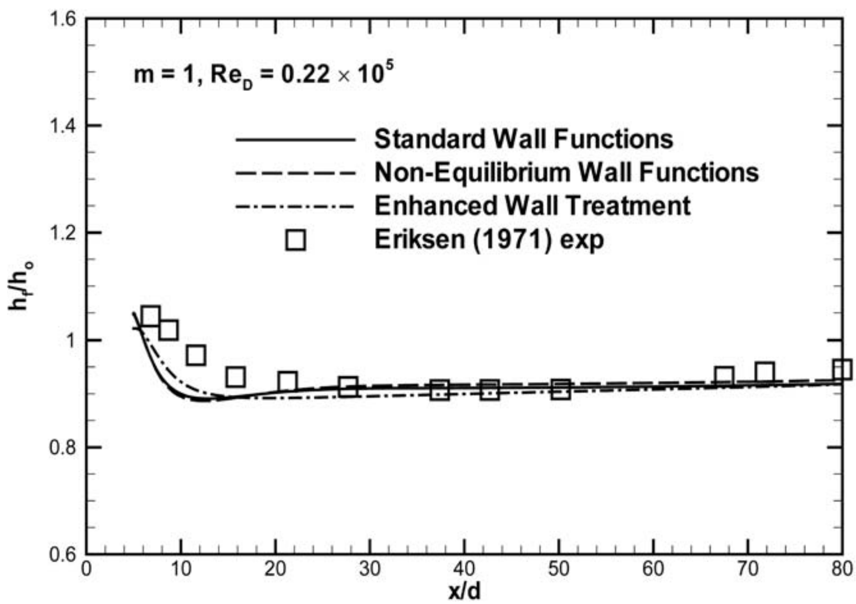

Figure 6.
The performance of different wall treatments with Eriksen (1971) experimental data

different wall treatments with the same k-ε model, namely the standard wall functions, the non-equilibrium wall functions, and the enhanced wall treatment – a two-layer-based wall function. Contrary to the commonly accepted notion that two-layer-based wall treatment gives better results, the three different wall treatments essentially yielded the same results provided that the near wall mesh requirements were met. The presence of the wall significantly affects turbulent flow field. In the near wall region, the solution variables such as velocities and temperatures have large gradients, and the accuracy of the final numerical solution primarily depends on how successfully the near wall boundary layers are modeled. Unfortunately, most current turbulence models, including the k-ε models, are only valid for turbulent core flows far from walls. In order to render these models suitable for wall-bounded flows, a series of empirical functions are introduced in order to resolve the boundary layers. The wall functions that bridge or link the solution variables at the near-wall cells and the corresponding quantities on the wall include formulas for mean velocity and temperature, as well as other scalars and formulas for near-wall turbulent quantities.

Thus, it has been demonstrated that k-ε model outperforms the other turbulence models. Therefore, the k-ε turbulence model has been chosen to carry out the simulations of the remaining cases. Since there is no experimental data available for the new scheme, shown in Figure 2, the dimension of the new scheme is scaled to be exactly the same as that of Sen's (1996) case. This means that at the same blowing ratio used in Sen's (1996) case, the same amount of coolant will pass through the jet hole of the new scheme. Thus, at the same blowing ratio with the same amount of coolant being used and with all the other parameters remaining the same, the performance of the new scheme will be first compared with that of Sen (1996), a circular jet.

Figure 7 compares the predictions of the centerline h_f/h_o of Sen's (1996) case and the new scheme with experimental data of Sen (1996). At $m = 1$, in Sen's (1996) scheme, the jets lift off from the testing surface, causing a substantial rise in heat transfer immediately after the injection, and the prediction agrees very well with the experimental data. In the new scheme at $m = 1$, the jets remain attached to the surface, therefore, the heat transfer is considerably lower than that of circular jets, especially in the near hole area of around $x/d = 5$, as shown in Figure 7(a), indicating better protection than traditional cylindrical holes. Beyond x/d of 15, due to turbulence, the diluted coolant dissipates into the mainstream and the h_f/h_o gradually increases towards one in the new scheme. For Sen's (1996) scheme, the jets reattach to the surface after a certain distance from the injection causing h_f/h_o to decrease slightly. At $m = 0.5$, the jets in both Sen's (1996) scheme and the new scheme remain attached to the protected surface. The heat transfer in the new scheme is slightly lower than that of the traditional cylindrical scheme in the near hole region and at the same level in the region far from the hole after x/d of 15 as shown in Figure 7(b).

The predictions of local heat transfer in the new scheme are shown in Figure 8. At $m = 1$, as shown in Figure 8(a), the new scheme gives a lower heat transfer coefficient than the traditional cylindrical hole due to the jet liftoff effect in the near hole region, especially in the area of $z/d < 0.8$. In the other area of $z/d > 0.8$, both schemes produce a heat transfer level of close to one. At $m = 0.5$, the h_f/h_o lingers around one since both schemes do not undergo jet liftoff, as shown in Figure 8(b).

Figure 9 compares the predictions which include conjugate heat transfer with those that do not. The effect of heat conduction within the test plate was considered by

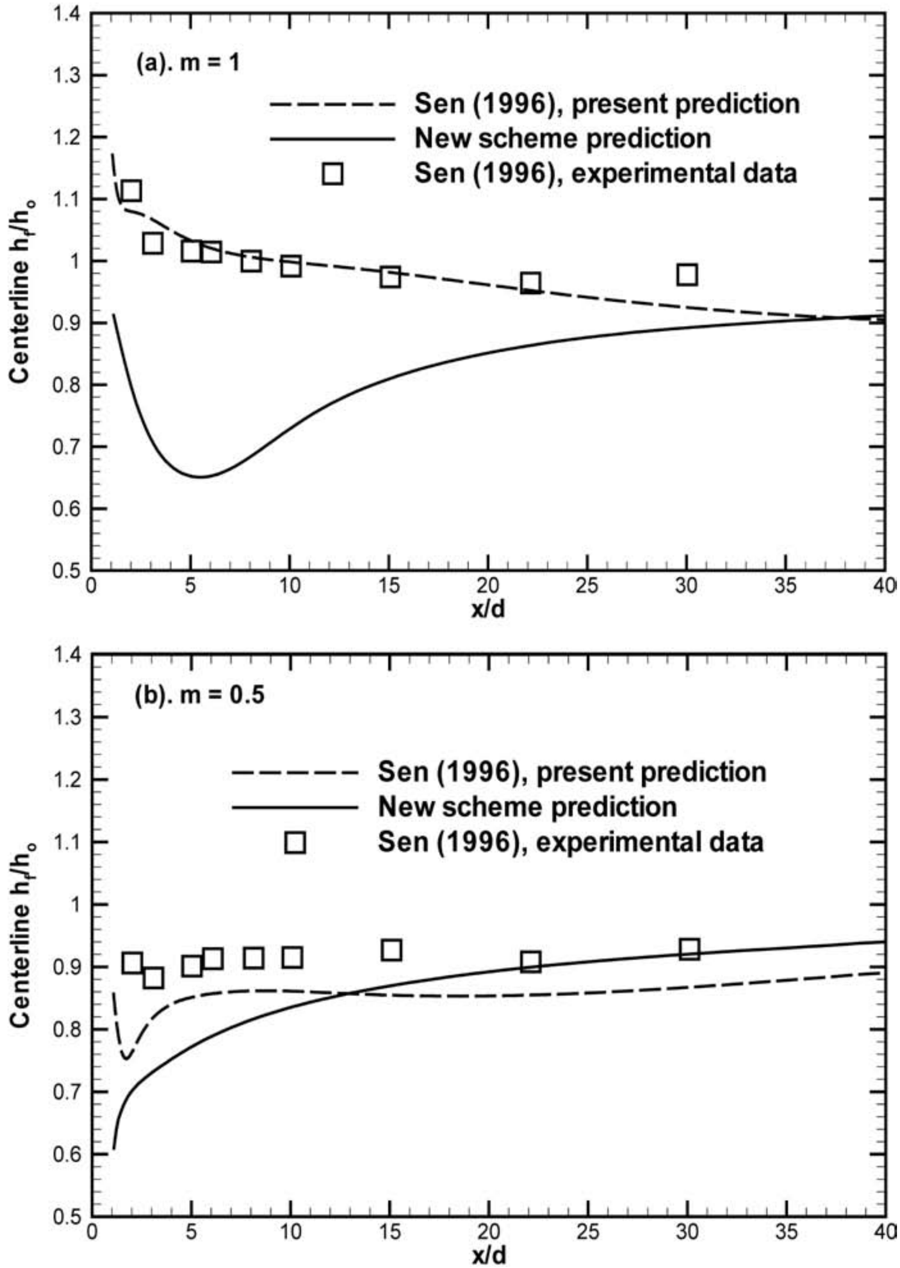

Figure 7.
Centerline *h* for the new scheme showing the new scheme giving lower *h* in the near hole region at high blowing ratio of 1

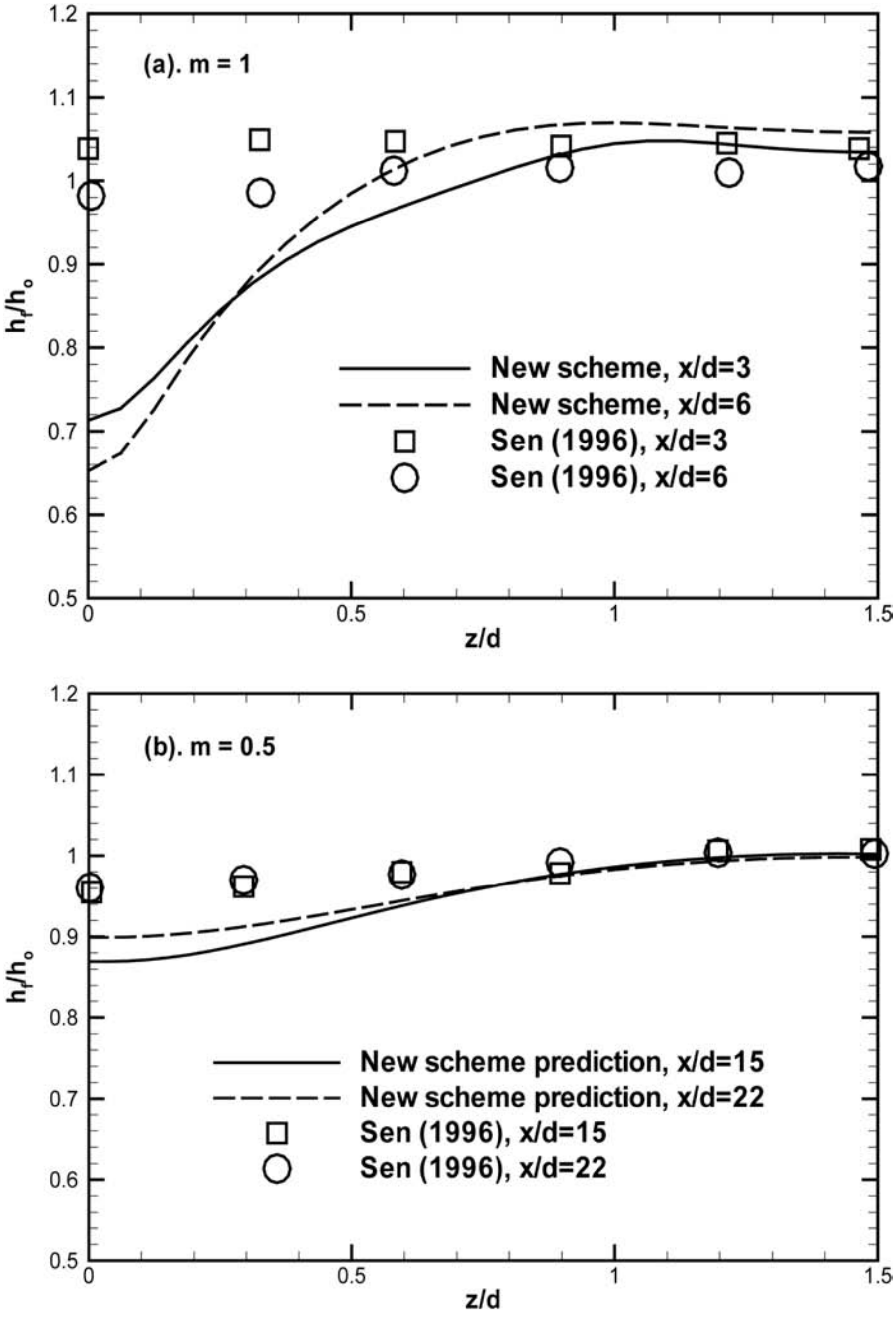

Figure 8.
Comparison of the local span-wise *h* for the new scheme with the standard scheme of Sen (1996)

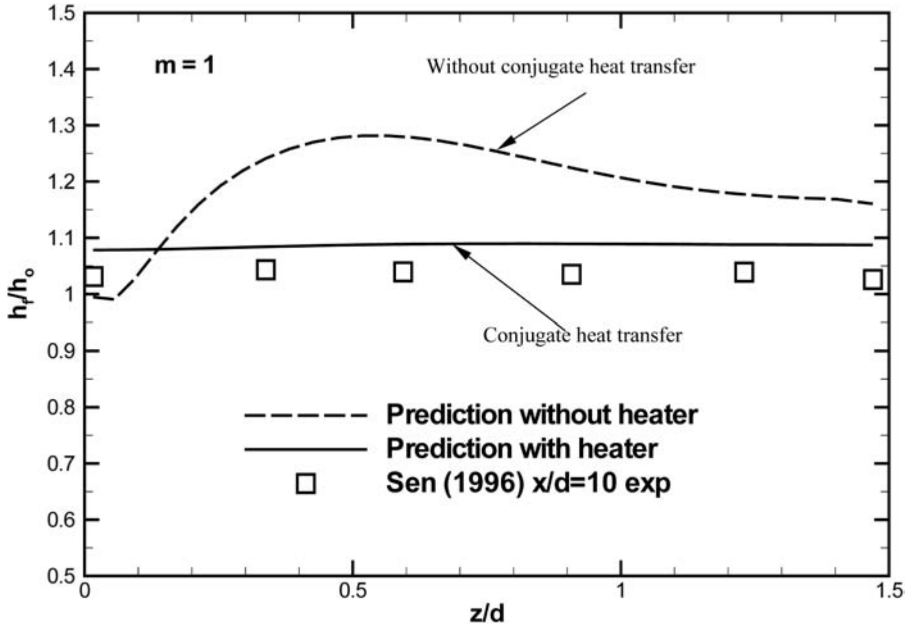

Figure 9.
The improvement on prediction of h in the local span-wise direction by modeling conjugate heat transfer (with heater)

modeling conjugate heat transfer between the main stream and the serpentine heating element with the computational domain including a finite thick foil downstream of the jet exits. The prediction without conjugate heat transfer shows that the heat transfer in the span-wise direction is over predicted around $z/d = 0.5$, and this can be attributed to heat conduction error in the test plate which has not been taken into account. Considerable improvement in the prediction of h_f/h_o in the span-wise direction is achieved when conjugate heat transfer was taken into account. On the other hand, there is little difference between them on the centerline heat transfer in the stream-wise direction (results not shown here). Thus, the thickness and the properties of the material, mainly thermal conductivity of the material of the heater, have been shown to affect the final solution in a no minor way. However, these values were not available in the work of Sen (1996). As a result, assumed value of heat conductivity (40 W/m K based on stainless steel foil in the experimental work) and foil thickness have been used in the simulations with conjugate heat transfer in order to tentatively obtain a solution.

It has been shown in the open literature that in the traditional cylindrical hole scheme, the gradient of adiabatic temperature in the span-wise direction is significantly larger than that of stream-wise direction. Therefore, there will be a huge driving force of heat conduction on the testing surface in the span-wise direction which tends to move energy from the high temperature end to the low temperature end. This heat conduction error in the span-wise direction causes the boundary condition of constant heat transfer on the testing surface to be no longer valid. As a result, the prediction in the span-wise direction deviates largely from the experimental data, especially in the area from $z/d = 0.2$ to 0.8 where a large temperature gradient exits. Again, the current turbulence models are unfairly being blamed for this disparity in the open literature.

As mentioned before, both the Bell *et al.* (2000) and Yu *et al.* (2002) used shaped holes with the same pitching ratio of 3. These geometries stand in the middle of the traditional circular jets and the new cooling scheme. Traditionally, the film cooling of cylindrical hole on a flat plate is the most fundamental problem of all film cooling, which has been studied extensively, both experimentally and numerically. Although it is the simplest cooling scheme, satisfactory numerical prediction is notoriously difficult to be obtained, particularly at high blowing ratio when jets undergo liftoff. In the shaped hole cases, satisfactory prediction is easier as the jets with expanded exits do not lift off causing flow separation and recirculation from the protected surfaces at higher blowing ratio. Figures 10 and 11 show the comparison between the predictions; by the present study with the experimental data by Bell *et al.* (2000) and Yu *et al.* (2002), respectively. An excellent agreement between predictions and experimental data is obtained. In Figure 11, the disagreement in the near hole region may be attributed to the heat conduction error in the experiment. These two cases further prove that the methodology developed in this study works very well with shaped holes. Therefore, the predictions of the new cooling scheme by this method are valid with certain confidence.

In comparison with the other recently published shaped holes shown in Figure 12, the lateral averaged heat transfer level of the new scheme at high blowing ratio is among the lowest along with the Bell *et al.* (2000) and Yu *et al.* (2002) schemes as shown in Figure 13(a). Both the Saumweber *et al.* (2003) and Dittmar *et al.* (2003) geometries are more like a slot which tends to have higher heat transfer than the cylindrical and fan-shaped hole results because of more disruption of boundary layer by more elongated jet exits in the span-wise direction. At a low blowing ratio, the difference of the heat transfer level between different schemes are not as apparent as at a higher blowing ratio

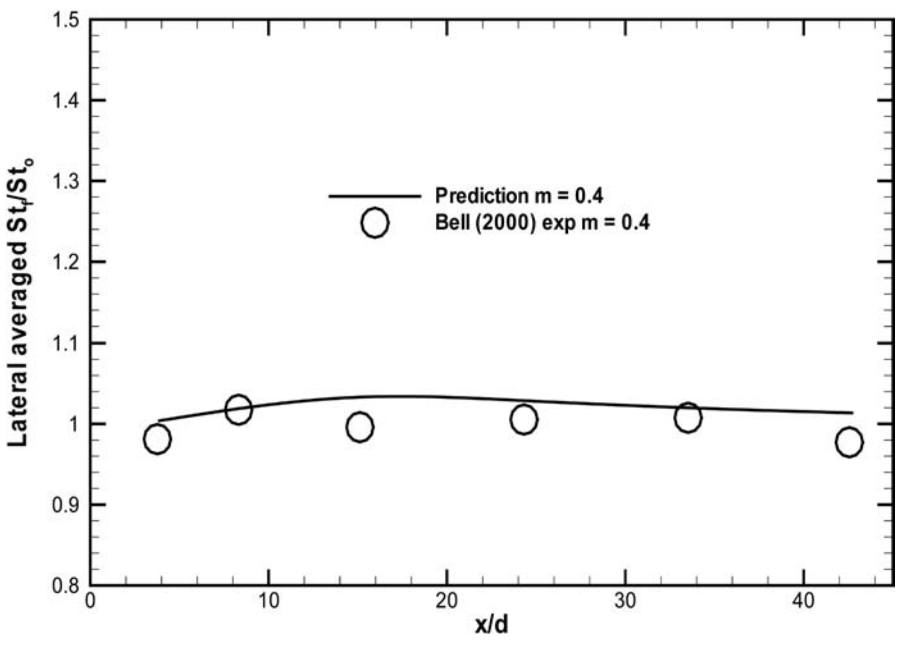

Figure 10.
Prediction of lateral averaged *St* showing the good agreement with the experimental data for Bell case

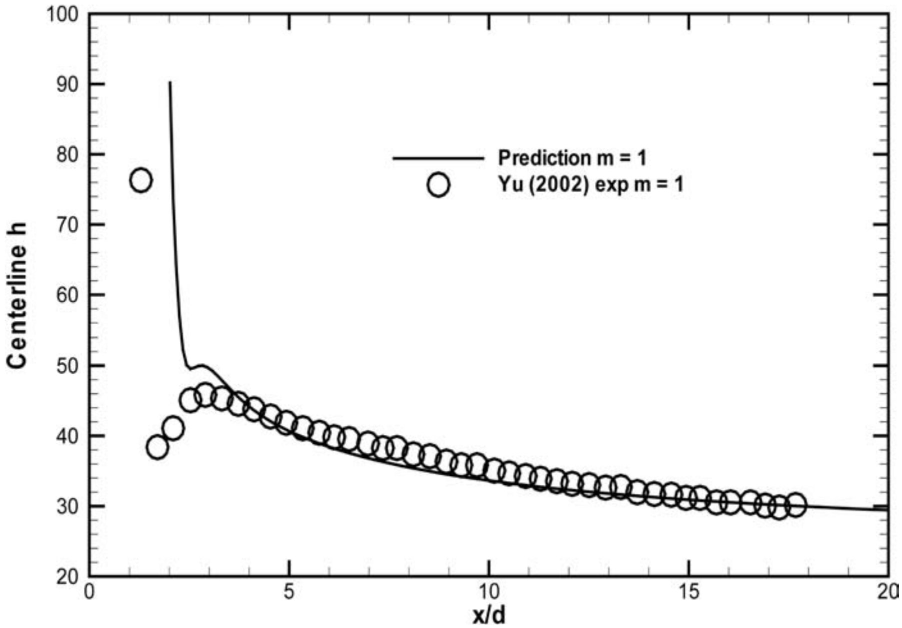

Figure 11.
Prediction of centerline h showing the good agreement with the experimental data for Yu case

due to the fact that reduced momentum of the jets in different schemes caused less penetration of the jets, as shown in Figure 13(b). Overall, all the three advanced cooling schemes – Bell *et al.* (2000), Yu *et al.* (2002), and the new scheme – give the same level of heat transfer to a certain degree. Another interesting fact about the new scheme is that the lateral averaged heat transfer is insensitive to the blowing ratio, a definite advantage considering that heat transfer of circular jet increases with blowing ratio. Figure 14 shows the comparison of lateral averaged adiabatic effectiveness between different cooling schemes, namely, traditional circular holes by Sinha *et al.* (1991), forward diffused holes by Bell *et al.* (2000), both forward and lateral shaped holes by Yu *et al.* (2002), straight far-shaped holes by Dittmar *et al.* (2003), far-shaped holes by Saumweber *et al.* (2003), and forward-lateral-diffused holes by Taslim and Khanicheh (2005), geometries shown in Figure 12. Obviously, the new scheme gives the highest lateral averaged effectiveness even at low blowing ratio below 0.5 when the jets in all schemes including the traditional cylindrical holes remain attached to the protected surface, as shown in Figure 14(a). The advantages of the new scheme are more evident at higher blowing ratio of above one when some cooling schemes such as circular holes undergo liftoff, as shown in Figure 14(b). After the coolant enters the bend, its momentum is reduced immediately as the cross-section of the flow path experiences a sudden increase. Further downstream toward the exit, the area of flow path increases gradually until the exit, further reducing the coolant momentum and enlarging the coverage area. All these factors contribute to the enhanced protection of the surface.

5. Conclusions

In the present study, an extensive investigation has been carried out to evaluate the performance of different turbulence models as well as the different wall treatments

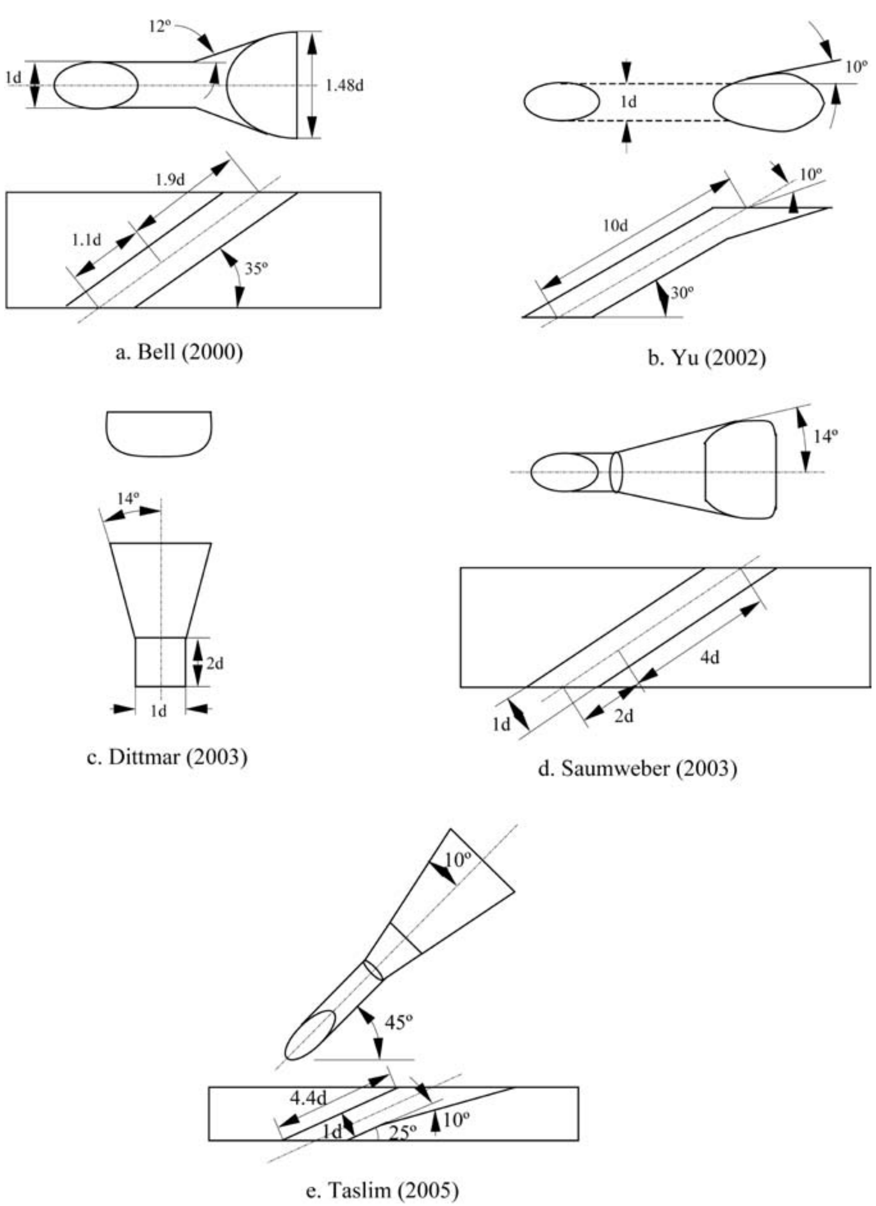

Figure 12.
Different cooling geometries (reproduced here) studied by researchers

in film cooling simulations. Systematic simulations of different benchmark cases have been performed and good agreement between the predictions and available experimental data has been shown. The methodology, fully established in those benchmark case simulations, is applied to a novel cooling scheme and its results are compared with those of the other shaped holes published in the literature in recent

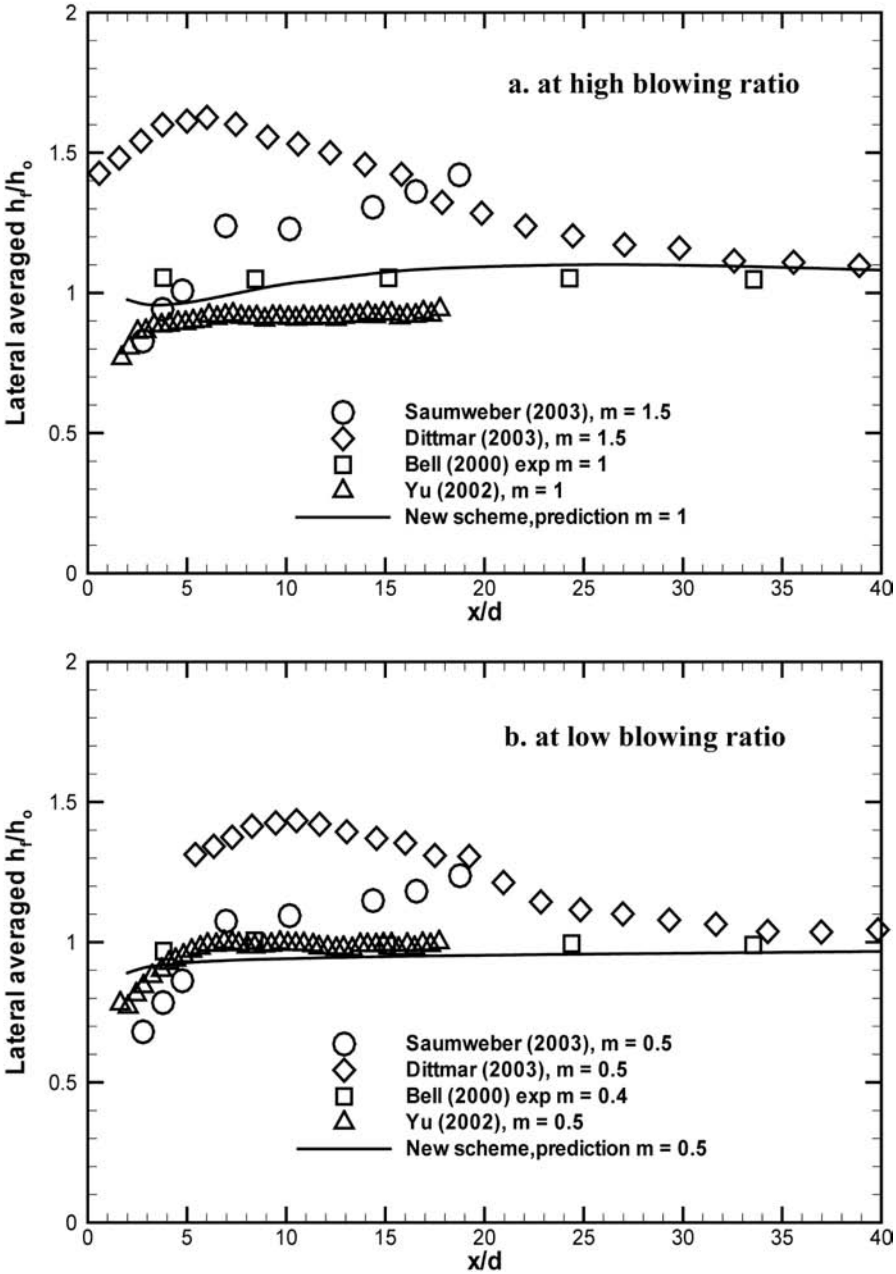

Figure 13.
Comparison of lateral averaged h between the new scheme and shaped holes

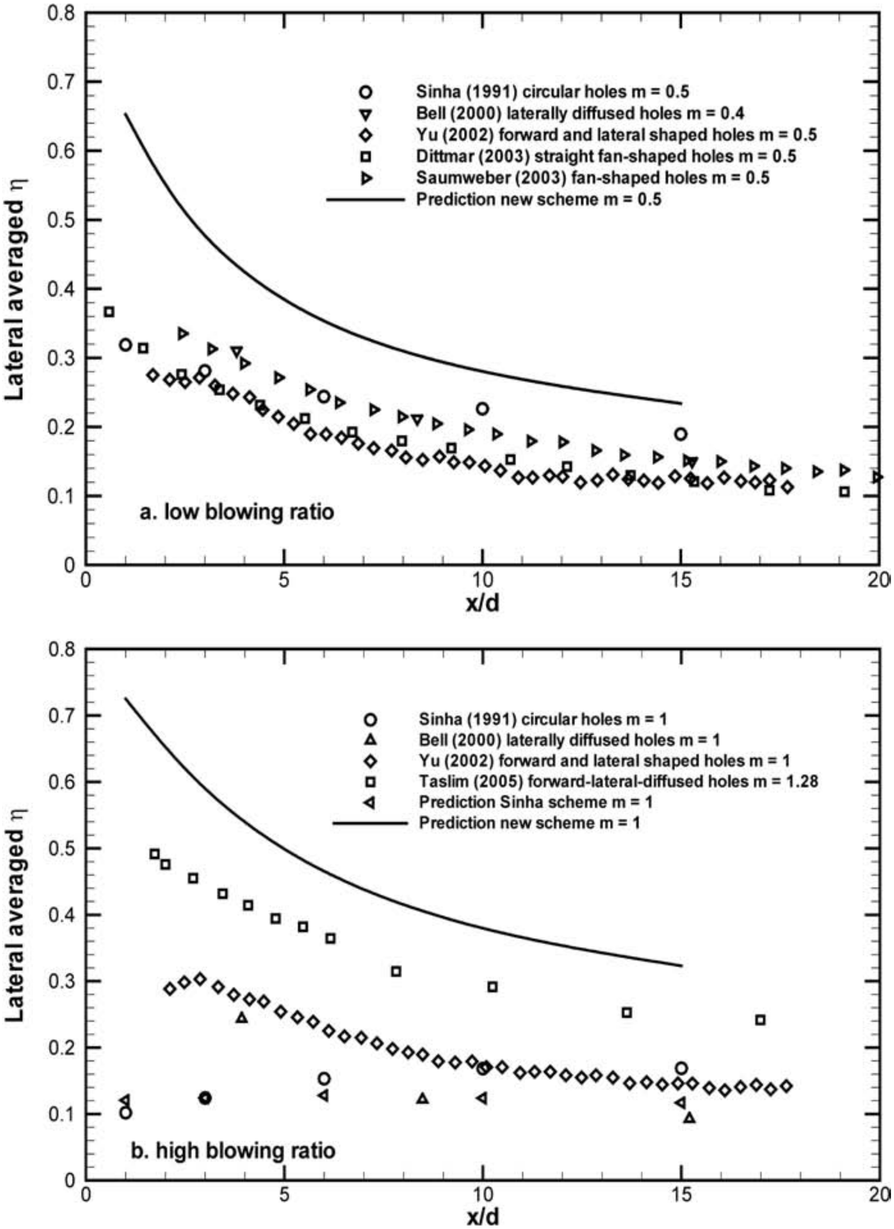

Figure 14.
Comparison of lateral averaged effectiveness between the new scheme and different cooling schemes

years. This study has shown that if used with care, CFD can be successfully employed with the current turbulence models in the prediction of the right levels of heat transfer. The following summarizes the findings in the present study:

- All the wall treatments, including standard wall functions, non-equilibrium wall functions, and two-layer zonal or enhanced wall treatment essentially give the same results as long as near wall mesh requirements are met, consistent with the physics, although two-layer zonal model has been shown to perform better by other researchers. This conclusion indicates that any effort intended for using two-layer zonal model or enhanced wall treatment to attain more accurate result will not meet much success from the beginning in view of the fact that all wall treatments are empirical functions and all empirical functions are very close to each other.
- Contrary to the consistency of different near wall treatments, unfortunately, the predictions of different turbulence models in the film cooling simulations vary considerably depending on which turbulence model is selected, indicating a lack of fundamental understanding of the physics of the phenomenon of turbulence in the current turbulence models. Before the perfect turbulence model is invented, engineers have to make the most of the current turbulence models with great caution, particularly in some applications where flow separation and recirculation occur.
- Conduction error on the test surface in the experiments intended for heat transfer coefficient measurement is considerably larger than believed by the technical community, which is why different researchers measured different levels of heat transfer coefficients from different structures of constant heat flux plate. Modeling conjugate heat transfer by considering the effect of heat conduction in the testing plate can gives more realistic result, especially in the span-wise direction.
- The proposed cooling scheme gives considerable lower heat transfer coefficient at the centerline in the near hole region than the traditional cylindrical hole, especially at a high blowing ratio when traditional cylindrical hole undergoes liftoff. Although the new scheme gives the same level of heat transfer as the other advanced cooling schemes, the new scheme produces much higher level of cooling effectiveness, 20-60 percent more than other schemes.

Since the reliability of the experimental data of heat transfer coefficient in the open literature is questionable, new experiments specifically intended for validating numerical models need to be done. Moreover, the performance of the new scheme on curved surfaces certainly will be different than that on the flat plate and should be studied in the future.

References

Ai, D., Ding, P.P. and Chen, P.H. (2001), "The selection criterion of injection temperature pair for transient liquid crystal thermography on film cooling measurements", *International Journal of Heat and Mass Transfer*, Vol. 44, pp. 1389-99.

Al-Hamadi, A.K., Jubran, B.A. and Theodoridis, G. (1998), "Turbulence intensity effects on film cooling and heat transfer from compound angle holes with particular application to gas turbine blades", *Energy Convers. Mgmt*, Vol. 39 No. 14, pp. 1449-57.

Ammari, H.D., Hay, N. and Lampard, D. (1990), "The effect of density ratio on the heat transfer coefficient from a film-cooled flat plate", *Journal of Turbomachinery*, Vol. 112, pp. 444-50.

Azad, G.S., Han, J.C., Teng, S. and Boyle, R. (2000), "Heat transfer and pressure distributions on a gas turbine blade tip", *ASME Journal of Turbomachinery*, Vol. 122, pp. 717-24.

Baldauf, S., Scheurlen, M., Schulz, A. and Wittig, S. (2002), "Heat flux reduction from film cooling and correlation of heat transfer coefficients from thermographic measurements at enginelike conditions", *Journal of Turbomachinery*, Vol. 124, pp. 699-709.

Bell, C.M., Hamakawa, H. and Ligrani, P.M. (2000), "Film cooling from shaped holes", *Journal of Heat Transfer*, Vol. 122, pp. 224-32.

Brittingham, R.A. and Leylek, J.H. (2000), "A detailed analysis of film cooling physics: part IV – compound – angle injection with shaped holes", *Journal of Turbomachinery*, Vol. 122, pp. 133-45.

Burd, S.W., Kaszeta, R.W. and Simon, T.W. (1998), "Measurements in film cooling flows: hole L/D and turbulence intensity", *ASME Journal of Turbomachinery*, Vol. 120, pp. 791-8.

Cho, H.H., Rhee, D.H. and Kim, B.G. (2001), "Enhancement of film cooling performance using a shaped film cooling hole with compound angle injection", *JSME International Journal, Series B*, Vol. 44 No. 1, pp. 99-110.

Cutbirth, J.M. and Bogard, D.G. (2002), "Thermal field and flow visualization within the stagnation region of a film-cooled turbine vane", *Journal of Turbomachinery*, Vol. 124, pp. 200-6.

Dittmar, J., Schulz, A. and Wittig, S. (2003), "Assessment of various film-cooling configurations including shaped and compound angle holes based on large-scale experiments", *Journal of Turbomachinery*, Vol. 125, pp. 57-64.

Ekkad, S.V., Han, J.C. and Du, H. (1998), "Detailed film cooling measurements on a cylindrical leading edge model: effect of free-stream turbulence and coolant density", *Journal of Turbomachinery*, Vol. 120, pp. 799-807.

Eriksen, V.L. (1971), "Film cooling effectiveness and heat transfer with injection through holes", PhD dissertation, University of Minnesota, Minneapolis, MN.

Gartshore, I., Salcudean, M. and Hassan, I. (2001), "Film cooling injection hole geometry: hole shape comparison for compound cooling orientation", *AIAA Journal*, Vol. 39 No. 8, pp. 1493-9.

Goldstein, R.J. and Jin, P. (2001), "Film cooling downstream of a row of discrete holes with compound angle", *Journal of Turbomachinery*, Vol. 123, pp. 222-30.

Gritsch, M., Schulz, A. and Wittig, S. (2001), "Effect of crossflows on the discharge coefficient of film cooling holes with varying angles of inclination and orientation", *Journal of Turbomachinery*, Vol. 123, pp. 781-7.

Gritsch, M., Schulz, A. and Wittig, S. (2003), "Effect of internal coolant crossflow on the effectiveness of shape film-cooling holes", *Journal of Turbomachinery*, Vol. 125, pp. 547-54.

Hale, C.A. (1999), "An experimental and numerical study of the hydrodynamics and surface heat transfer associated with short film cooling holes fed by a narrow plenum", PhD dissertation, Purdue University, West Lafayette, IN.

Harrington, M.K., McWaters, M.A., Bogard, D.G., Lemmon, C.A. and Thole, K.A. (2001), "Full-coverage film cooling with short normal injection holes", *Journal of Turbomachinery*, Vol. 123, pp. 798-805.

Hyams, D.G. and Leylek, J.H. (2000), "A detailed analysis of film-cooling physics: part III – streamwise injection with shaped holes", *Journal of Turbomachinery*, Vol. 122, pp. 122-32.

Immarigeon, A.A. (2004), "Advanced impingement/film-cooling schemes for high-temperature gas turbine – numerical study", MSc thesis, Concordia University, Montreal.

Jung, I.S. and Lee, J.S. (2000), "Effects of orientation angles on film cooling over a flat plate: boundary layer temperature distributions and adiabatic film cooling effectiveness", *Journal of Turbomachinery*, Vol. 122, pp. 153-60.

Kim, Y.J. and Kim, S.M. (2004), "Influence of shaped injection holes on turbine blade leading edge film cooling", *International Journal of Heat and Mass Transfer*, Vol. 47, pp. 245-56.

Kohli, A. and Bogard, D.G. (1999), "Effects of hole shape on film cooling with large angle injection", ASME Paper 99-GT-165.

Li, X., Gaddis, J.L. and Wang, T. (2001), "Mist/steam heat transfer in confined slot jet impingement", *Journal of Turbomachinery*, Vol. 123, pp. 161-7.

Maiteh, B.Y. and Jubran, B.A. (1999), "Influence of mainstream flow history on film cooling and heat transfer from two rows of simple and compound angle holes in combination", *International Journal of Heat and Fluid Flow*, Vol. 20, pp. 158-65.

Mayhew, J.E., Baughn, J.W. and Byerley, A.R. (2003), "The effects of free-stream turbulence on film cooling adiabatic effectiveness", *International Journal of Heat and Fluid Flow*, Vol. 24, pp. 669-79.

Nasir, H., Acharya, S. and Ekkad, S. (2003), "Improved film cooling from cylindrical angled holes with triangular tabs: effect of tab orientations", *International Journal of Heat and Fluid Flow*, Vol. 24, pp. 657-68.

Rhee, D.H., Lee, Y.S. and Cho, H.H. (2002), "Film cooling effectiveness and heat transfer of rectangular-shaped film cooling holes", ASME Paper No. GT-2002-30168.

Sargison, J.E., Guo, S.M., Oldfield, M.L.G., Lock, G.D. and Rawlinson, A.J. (2002), "A converging slot-hole film-cooling geometry – part I: low-speed flat-plate heat transfer and loss", *Journal of Turbomachinery*, Vol. 124, pp. 453-60.

Saumweber, C., Schulz, A. and Wittig, S. (2003), "Free-stream turbulence effects on film cooling with shaped holes", *Journal of Turbomachinery*, Vol. 125, pp. 65-72.

Schmidt, D.L., Sen, B. and Bogard, D.G. (1996), "Effects of surface roughness on film cooling", ASME Paper 96-GT-299.

Sen, B. (1996), "Effect of injection hole geometry, high freestream turbulence, and surface roughness on film cooling heat transfer", PhD dissertation, The University of Texas, Austin, TX.

Sinha, A.K., Bogard, D.G. and Crawford, M.E. (1991), "Film-cooling effectiveness downstream of a single row of holes with variable density ratio", *Journal of Turbomachinery*, Vol. 113, pp. 442-9.

Son, C., Gillespie, D., Ireland, P. and Dailey, G. (2001), "Heat transfer and flow characteristics of an engine representative impingement cooling system", *Transactions of the ASME*, Vol. 123, pp. 154-60.

Taslim, M.E. and Khanicheh, A. (2005), "Film effectiveness downstream of a row of compound angle film holes", *ASME Journal of Heat Transfer*, Vol. 127, pp. 434-40.

Taslim, M.E., Setayeshgar, L. and Spring, S.D. (2001), "An experimental evaluation of advanced leading edge impingement cooling concepts", *ASME Journal of Turbomachinery*, Vol. 123, pp. 147-53.

Vogel, G., Graf, A.B.A., Wolfersdorf, J.V. and Weigand, B. (2003), "A novel transient heater-foil technique for liquid crystal experiments on film-cooled surfaces", *Journal of Turbomachinery*, Vol. 125, pp. 529-37.

Wilfert, G. and Wolff, S. (2000), "Influence of internal flow on film cooling effectiveness", *Journal of Turbomachinery*, Vol. 122, pp. 327-33.

York, W.D. and Leylek, J.H. (2003), "Leading-edge film-cooling physics – part III: diffused hole effectiveness", *Journal of Turbomachinery*, Vol. 125, pp. 252-9.

Yu, Y., Yen, C.H., Shih, T.I.P. and Chyu, M.K. (2002), "Film cooling effectiveness and heat transfer coefficient distributions around diffusion shaped holes", *Journal of Heat Transfer*, Vol. 124, pp. 820-7.

Yuen, C.H.N. and Martinez-Botas, R.F. (2003), "Film cooling characteristics of a single round hole at various streamwise angles in a crossflow: part I effectiveness", *International Journal of Heat and Mass Transfer*, Vol. 46, pp. 221-35.

Further reading

Ferguson, J.D., Walters, D.K. and Leylek, J.H. (1998), "Performance of turbulence models and near-wall treatments in discrete jet film cooling simulations", ASME Paper 98-GT-438.

Leylek, J.H. and Zerkle, R.D. (1993), "Discrete-jet film cooling: a comparison of computational results with experiments", ASME Paper 93-GT-207.

Walters, D.K. and Leylek, J.H. (2000), "A detailed analysis of film cooling physics: part I: streamwise injection with cylindrical holes", *Journal of Turbomachinery*, Vol. 122, pp. 102-12.

Corresponding author

I. Hassan can be contacted at: IbrahimH@alcor.concordia.ca