NUMERICAL MODELING OF A TURBULENT SEMI-CONFINED SLOT JET IMPINGING ON A CONCAVE SURFACE

by

Hadi AHMADI [a], Mehran RAJABI-ZARGARABADI [a*], Arun S. MUJUMDAR [b], and Javad MOHAMMADPOUR [a]

[a] Department of Mechanical Engineering, Semnan University, Seman, Iran
[b] Department of Mechanical Engineering, National University of Singapore, Singapore

Original scientific paper
DOI: 10.2298/TSCI121003057A

This article presents results from a numerical study of a turbulent slot jet impinging on a concave surface. Five different low Reynolds number k-ε models were evaluated to predict the heat transfer under a two dimensional steady turbulent jet. The effects of flow and geometrical parameters (e. g. jet Reynolds number and jet-to-target separation distance) have been investigated. The Yap correction is applied for reducing the over-prediction of Nusselt number in the near wall region. It is shown that among the models tested in this study, the LS-Yap model is capable of predicting local Nusselt number in good agreement with the experimental data in both stagnation and wall jet region. Moreover, after implementation of Yap correction, no significant effect of the nozzle-to-surface distance, h/B, on the predicted stagnation Nusselt number has been found. Finally it is demonstrated that the higher values of turbulent Prandtl number reduces the heat diffusion along the wall and consequently the predicted local Nusselt number is reduced especially in the wall jet region.

Key words: *Nusselt number, low-Reynolds turbulence models, turbulent Prandtl number*

Introduction

Among all the convective cooling methods, impinging jet heat transfer produces the highest convection coefficient. So the use of impinging jet is a good technique for transferring heat to or from surfaces that need to be heated, cooled or dried.

Some applications of this technique include cooling of electronic devices, cutting and forming processes, drying of paper, textiles, woods, foods and film materials [1, 2]. The jet impinging on concave surface is an efficient approach for the removal of high heat transfer encountered in leading edge of turbine blades. Also, airfoil leading edge heating for anti-icing applications is another important application of this method, which is very useful in aerospace industries [3-5].

Most of the general principles of impinging flows are the same for both the flat and curved surfaces, but the flow physics of jets impinging on concave surfaces, as compared to flat surfaces, undergoes many changes.

* Corresponding author; e-mail: rajabi@semnan.ac.ir

One of the most important factors in jet impingement is surface curvature, which itself can create some other important effects due to centrifugal and Coriolis forces and can lead to enhanced wall jet development in the circumferential direction. Surface curvature also has a strong effect on turbulent boundary layer development. In recent studies carried out on concave surfaces, notable improvement in the heat transfer rate has been reported relative to that to flat surfaces [4]. Numerous investigations have been performed concerning impinging jet flow and heat transfer for flat surfaces. Although the analysis of jet flows over flat surfaces is important, the importance of curved surfaces with their many applications in various industries should not be ignored.

Over the past several decades, extensive studies have also been conducted on heat and mass transfer characteristics in impinging jet on concave surfaces [3-10]. Choi *et al*. [6] experimentally investigated a single slot-jet impinging on a semi-circular concave surface at different jet-to-target distances and jet Reynolds numbers, and evaluated the effect of this impingement on wall Nusselt number and hydrodynamic characteristics. Kayansayan and Kucuka [9] performed an experimental study on the cooling effect induced by a confined slot jet impinging on a semi-circular channel. They presented local Nusselt number distributions on this surface at jet Reynolds numbers ranging from 200 to 11000 and jet-to-target distances ranging from 2.2 to 4.2. Eren *et al*. [7] experimentally investigated the non-liner flow and heat transfer characteristics caused by the impingement of a slot jet on a concave surface. In this study, for a fixed nozzle-to-surface distance, the impacts of different Reynolds numbers on jet velocity distribution and Nusselt number changes were reviewed and ultimately, correlation were derived for the calculation of local, stagnation point, and average Nusselt numbers as a function of jet Reynolds number and dimensionless jet-to-target distance.

Hosseinalipour and Mujuamdar [11] carried out a numerical investigation to predict the fluid flow and heat transfer characteristics of 2-D turbulent confined impinging jet flow on flat surface and opposing jet flow. Five low Reynolds number k-ε models and the standard high-Reynolds-number model were used in the simulation. Also, the effect of Yap correction was tested on low Reynolds models and it was found that in some models this correction improves the heat transfer predictions. Seyedein *et al*. [12] numerically presented the results of simulation of two-dimensional flow field and heat transfer impingement due to a turbulent single heated slot jet discharging normally into a confined channel by using both low-Reynolds and high-Reynolds number versions of k-ε turbulence models. It was found that Launder and Sharma (LS) and Lam and Bremhorst (LB) models show very good agreement with the experimental data for the prediction of heat transfer distribution. Souris *et al.* [13] applied the two-equation turbulence k-ε model and the Reynolds stress model (RSM) to numerically simulate the cooling performed by jet impingement on a semi-circular concave surface. They compared the numerical results with the experimental data of Choi *et al*. [6], which indicated good agreement between the predictions of both models and experimental data. Frageau *et al*. [5] calculated the flow and heat transfer due to an array of circular jets impinging on concave surface for anti-icing application of aircraft wing leading edge.

Wang and Mujumdar [14] investigated the impingement of a slot jet on a flat surface using five low-Reynolds k-ε turbulence models, and compared their results with published experimental data. The combination of low-Reynolds k-ε models with Yap correction was proposed for the reduction of turbulence length scale in the near-wall region. They found that for most of the tested models, it is capable of improving the predicted local Nusselt number in good agreement with the experimental data in both the stagnation and wall jet regions. Employing the k-ω SST turbulence model, Kumar and Prasad [15] numerically studied the flow and heat transfer resulting from a row of jets with circular cross-sections impinging on a semi-cylindrical concave surface. They reported the effects of jet Reynolds number, jet-to-target distance and nozzle-to-nozzle distance on the distribution of pressure and primary and secondary Nusselt number peaks on the target surfaces. According to the obtained results, with the reduction of jet-to-target surface and nozzle-to-nozzle distances and with the increase of jet Reynolds number, the rate of heat transfer increases.

The performances of several turbulence models in the prediction of convective heat transfer due to the impingement of a slot-jet onto flat and concave cylindrical surface were evaluated by Sharif and Mothe [3]. They considered five turbulence models in this study, including the standard k-ε model, k-ε RNG model, realizable k-ε model, k-ω SST model, and the Reynolds stress transport model. It has been shown that the predicted local Nusselt numbers obtained using both the k-ε RNG and k-ω SST models are in good agreement with the experimental data in [6]. However, by comparing the mean velocity profile on jet centerline with the experimental data, the k-ε RNG model was selected as the best model for predicting the local Nusselt number. Sharif and Mothe [4] numerically investigated the effect of Reynolds number and eventually derived a correlation for the estimation of average Nusselt number on concave surfaces as a function of the parameters considered in the study.

The flow field and heat transfer characteristics of a turbulent slot jet impinging on a semi-circular concave surface with uniform heat flux were studied by Yang *et al.* [16]. The standard k-ε model was performed in this case and the obtained results were compared with the experimental data. They reasonably predicted the local Nusselt number with a maximum discrepancy within 15%. They also reported that the effect of impingement distances on the average Nusselt number is not significant at high values of h/B. Bazdidi-Tehrani *et al.* [17] investigated the prediction of flow and heat transfer characteristics in the turbulent axisymmetric impinging jet by using both the v^2-f and algebraic turbulent heat flux models. Their results indicated that applying the higher order algebraic turbulent heat flux models, namely, generalized gradient diffusion hypothesis (GGDH) and HOGGDH (higher order GGDH) [17], results in larger wall jet heat diffusion and improves the predicted local Nusselt number in both the stagnation and wall jet regions.

In the present study, the performances of low-Reynolds k-ε models in the prediction of the Nusselt number of a slot-jet impinging on a concave surface are investigated. The results are compared with the available experimental data. The effect of Yap correction on the improvement of the predicted local Nusselt number is clearly demonstrated. Also a detailed review of effects of the important parameters on distribution of local Nusselt number is presented using the low-Reynolds k-ε models and Yap correction. Finally, the effect of turbulent Prandtl number on the distribution of local Nusselt number is investigated.

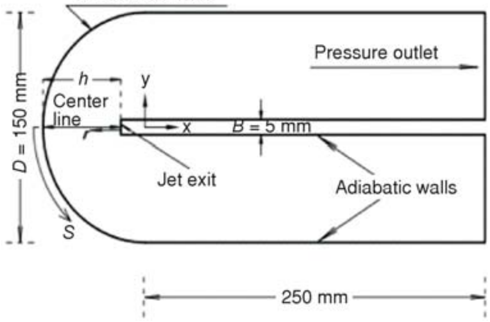

Figure 1. Schematic of slot-jet impingement on a concave surface and boundary conditions

Mathematical formulation

Governing equations

Figure 1 shows a schematic diagram of the geometry and computational domain of present work. The geometrical dimensions in this problem have been chosen identical to the experimental study of Choi *et al.* [6]. In this problem, a steady and turbulent 2-D flow is considered. It is also assumed that the working fluid (air) is Newtonian and incompressible. Due to the flow field symmetry, only half of domain is considered for calculations.

In view of the above assumptions, the governing equations of the problem are the continuity, momentum, and energy equations and the transport equations for the turbulent kinetic energy and dissipation added through the turbulence model:

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{1}$$

$$\rho U_i \frac{\partial U_j}{\partial x_i} = -\frac{\partial P}{\partial x_j} + \frac{\partial}{\partial x_i}\left[\mu\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) - \rho\overline{u_i' u_j'}\right] \tag{2a}$$

$$-\rho\overline{u_i' u_j'} = \mu_\mathrm{t}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) - \frac{2}{3}\delta_{ij}\rho k \tag{2b}$$

$$\rho c_p U_i \frac{\partial T}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\lambda\frac{\partial T}{\partial x_i} - \rho\overline{u_i' T'}\right] \tag{3}$$

Turbulence model

By considering the wide acceptability of k-ε models and also the good accuracy of low-Reynolds k-ε models in prediction of flow and heat transfer field due to jet impingement on surface [14, 18], these turbulence models have been applied in the present analysis. In the two-equation (k and ε) models, turbulence stress is related to velocity gradients and turbulence viscosity through Boussinesq approximation:

$$\rho U_i \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu\frac{\mu_\mathrm{t}}{\sigma_\mathrm{k}}\right)\frac{\partial k}{\partial x_i}\right] + \mu_t\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\frac{\partial U_i}{\partial x_j} - \rho(\varepsilon + D) \tag{4}$$

$$\rho U_i \frac{\partial \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu\frac{\mu_\mathrm{t}}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right] + f_1 C_1 \mu_\mathrm{t}\frac{\varepsilon}{k}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\frac{\partial U_i}{\partial x_j} - \rho f_2 C_2 \frac{\varepsilon^2}{k} + E \tag{5}$$

$$\mu_\mathrm{t} = \rho f_\mu C_\mu \frac{k^2}{\varepsilon} \tag{6}$$

where C_1, C_2, C_μ, σ_k, and σ_ε are constants of the equations and f_1, f_2, and f_μ are the damping functions of the low-Reynolds k-ε models. By means of the damping functions, it is possible to use low-Reynolds models in the near-wall regions. Five low Reynolds k-ε models applied in the present work, have been developed by Abid [19], Lam and Bremhorst [20], Launder and Sharma [21], Abe *et al.* [22], and Change *et al.* [23, 24]. These models thereafter referred to as (AB), (LB), (LS), (AKN), and (CHC), respectively. The values of all the constants and damping functions related to all five low-Reynolds k-ε models have been given in [19-24] and the numerical study of Wang and Mujumdar [14]; so they are not repeated here.

The simplest model used for heat flux vector is the simple eddy diffusivity (SED) model, which is a first-order model, and is defined as follows:

$$-\overline{u_i' T'} = \frac{\nu_\mathrm{t}}{\mathrm{Pr}_\mathrm{t}}\frac{\partial T}{\partial x_i} \tag{7}$$

The turbulent Prandtl number is usually taken as constant and commonly assumed to be equal to 0.85 for near-wall flows [25].

Problem description

Geometrical specifications

The geometrical specifications used in this problem are:
the width of the inlet nozzle (B) has been considered equal to 5 mm; the target surface has been assumed to be semi-circular with a diameter of 150 mm, has negligible thickness, and is stationary; a constant heat flux of 5000 W/m^2 is applied to the concave surface; three nozzle-to-surface distances in the form of h/B = 4, 6, and 10 and two Reynolds numbers of 4740 and 2960 have been considered; in the calculations, the inlet turbulence intensity has been specified as 5% and the hydraulic diameter is twice the diameter of jet nozzle ($2B$).

Fluid properties

Table 1 shows the thermo-physical properties of air. Except for density, which was specified by the equation of state of an incompressible ideal gas, the other fluid thermo-physical properties are assumed to be constant.

Table 1. Thermo-physical properties of air

Properties	Air
Dynamic viscosity, μ	$1.7894\cdot10^{-5}$
Thermal conductivity, λ	0.0242
Specific heat, c_p	1006.43
Prandtl number	0.744

Boundary conditions

The boundary conditions applied in the present work are illustrated in fig. 1. As shown in this figure, uniform velocity at constant temperature (298 K) is assumed at jet exit. The thermal boundary condition at the concave surface is a constant heat flux in accordance with the experiments [6] while the other walls are insulated. No-slip condition has been applied at all walls. Also at flow outlet, the pressure-outlet boundary condition has been assumed.

Numerical solution

Mass, momentum, and energy conservation equations have been discretized by the control volume technique. The pressure-velocity coupling has been established through the SIMPLE algorithm [26]. The first-order upwind method and central difference method have been used for calculating the convective and diffusive terms, respectively. Due to the use of low-Reynolds models in this problem, it is necessary to ensure that $y^+ < 1$ all along the concave wall. As shown in fig. 2, the process of grid independency comparing the convergence of the local Nusselt number variations along the concave surface has been performed for the case of h/B = 4, D/B = 30, and Re = 4740. A close examination of the plots reveals that, mesh configurations finer than those with 70,000 cells produce no significant change in the Nusselt number distribution. Also for other geometries, calculations similar to those have been carried out.

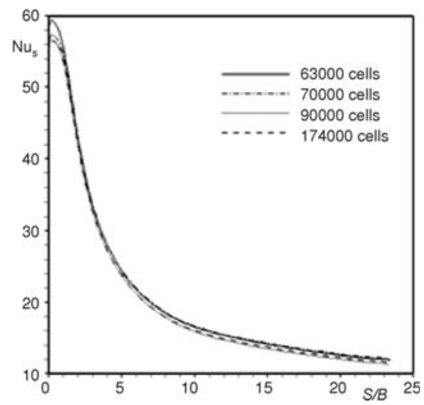

Figure 2. Effect of variation of grid dimensions on local Nusslet number

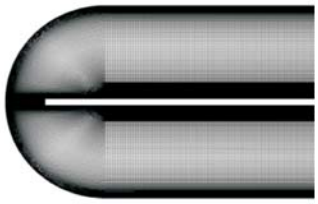

Figure 3. Structured grid used in the computations

The flow domain (see fig. 3) is a non-uniform grid system with the considered structure. In this numerical solution, convergence is obtained when the absolute residuals are be less than 10^{-6} and the temperature of the stagnation point remains constant with iteration.

Results and discussion

The performance of low-Reynolds *k-ε* models in the prediction of Nusselt number on concave surfaces has been evaluated for two different cases. Zero and non-zero Yap correction terms have been assumed for present work, respectively. The numerical results are compared with the experimental data.

Prediction of the Nusselt number by low-Reynolds k-ε models

In fig. 4, the results of comparison between five low-Reynolds *k-ε* models and the experimental data [6] at three different jet to target spaces is presented. According to this figure, in all three cases, large Nusselt number gradient changes are observed in the impingement region. Also, all the five models over-predict the local Nusselt number at the stagnation zone and under-predict this parameter in the wall jet region, in comparison with the experimental data. A close examination shows that for $h/B = 6$, some models predict the stagnation Nusselt number higher than those predicted for $h/B = 4$ and 10. Among these five models, only the LS model predicts results close to experimental data in the wall jet region ($S/B > 5$). However in the impingement region ($0 < S/B < 5$), the results obtained by this model are greater than the experimental results. By closely examining the results presented in fig. 4 it can be concluded that, as the distance between the nozzle and target surface increases, the numerical calculations predict the local Nusselt number distributions which are in better agreement with the available experimental data. It should be mentioned that based on experimental study [6] the uncertainty of Nusselt number is 7.94%.

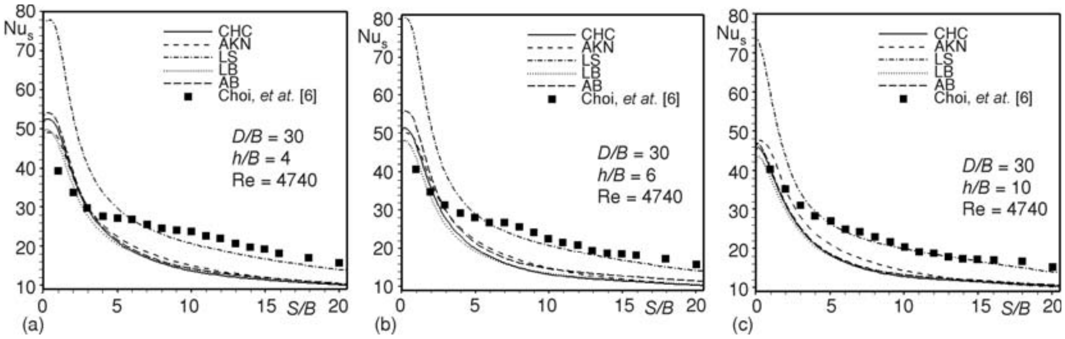

Figure 4. Predicted local Nusselt number distribution along the concave surface

Effect of Yap correction on the prediction of local Nusselt number

Launder [27] showed that there is a source term in the dissipation equation (*ε*), which has been assumed as zero in most of the previous studies. Yap [28] calculated this term as a function of the ratio of computational length scale to the local equilibrium length scale, and presented it as follows:

$$S_\varepsilon = \max\left[0.83\left(\frac{l}{l_\varepsilon}-1\right)\left(\frac{l}{l_\mathrm{e}}\right)^2\frac{\varepsilon^2}{k},\ 0\right] \tag{8}$$

By considering the important role of the turbulence dissipation rate in the k-ε models and also because of the existence of intensive velocity gradients and turbulent stresses which are created by jet impingement in stagnation zone, the expected turbulent kinetic energy is over-predicted in this region [11]. According to Yap [28], since the turbulence length scale is much larger than the near wall equilibrium length scale ($l > l_\mathrm{e}$) in these conditions, the source term becomes positive and $(l/l_\mathrm{e})^2$ allows the required increases of value of ε. So adding the Yap correction term to ε equation leads to control of dissipation rate and subsequently corrects the estimation of turbulent kinetic energy. Since the values of k directly affect the heat transfer rate [29], the prediction of local Nusselt number significantly improves in impingement zone. By moving along the circumferential direction and decreasing the flow turbulent intensity especially near the wall, the rate of turbulent kinetic energy (k) and turbulence dissipation (ε) are diminished. So the turbulence length scale is approached to near-wall equilibrium length scale ($l = l_\mathrm{e}$) which vanishes the source term and leads to considerable reduction of effect of Yap correction in wall jet region.

Figure 5 illustrates the effect of applying the Yap correction term into the dissipation equation (ε) for the prediction of local Nusselt number on a concave surface by the five low Reynolds number k-ε models. In comparison with the case without the Yap correction the results significantly improve in the impingement zone; however in the wall jet region, no change is observed. It is also found that the results obtained from the LS-Yap model, in comparison with the RNG model (investigated in previous research works by Sharif *et al.* [3]) and other low Reynolds models, are in a good agreement with the experimental data all along the wall length. According to fig. 5, by increasing the nozzle distance from the target surface, no significant enhancement of the local Nusselt number distribution is found. However, a better agreement with experimental data is achieved at higher values of *h/B*.

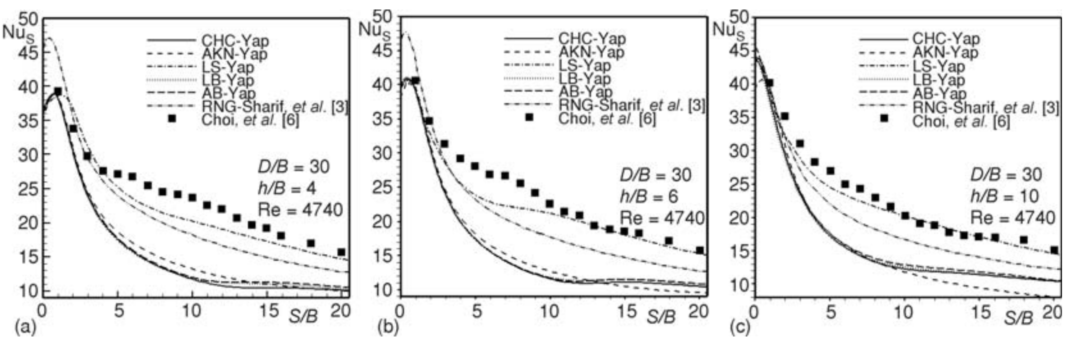

Figure 5. Effect of Yap correction on the prediction of local Nusselt number in comparison with experimental data [6]

Jet centerline velocity distribution

The variation of the non-dimensional velocity along the centerline (along the axial distance between jet and target surface) performed by the LS-Yap model is illustrated in fig. 6. The predicted results through the LS-Yap model, in comparison with the RNG model [3], have better agreement with the experimental data [6]. Due to the sensitivity at the near-wall region, velocity changes in this region have been more closely examined in magnified view. In addition to the above results, fig. 6 shows applying the Yap correction leads to under prediction of velocity near the wall region. The reduction of predicted velocity consequently improves the over prediction of Nusselt number of the impingement region and better agreements with experimental data are achieved (see fig. 5).

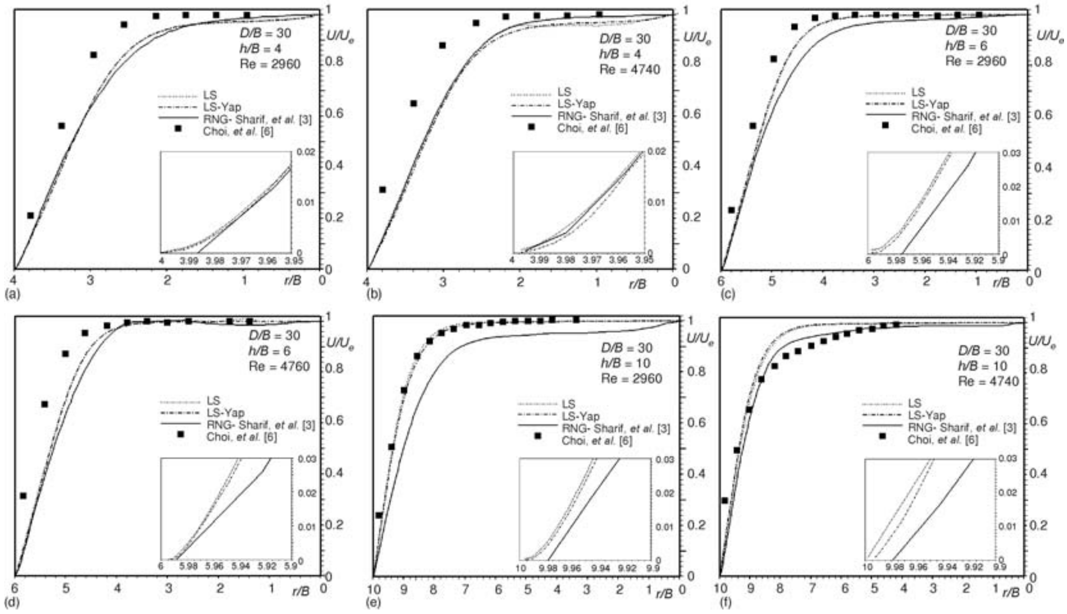

Figure 6. Effect of Yap correction on the predicted mean axial velocity

According to figs. 5 and 6, since the results obtained by the LS-Yap model have better agreement with the experimental results (in comparison with other turbulence models), this model can be considered as a suitable model for the prediction of heat transfer on concave surfaces.

Evaluation of turbulent kinetic energy

Figure 7 illustrates the turbulent kinetic energy distributions for the case of D/B = 30, Re = 2960 and 4740, and h/B = 6 and 10, which have been obtained by means of the LS-Yap model. It can be observed that with the reduction of jet Reynolds number and the increase of nozzle-to-surface distance, the quantity of turbulent kinetic energy diminishes; while the impact of Reynolds number variations is more considerable. Also the variation of velocity and consequently turbulent kinetic energy near the target wall significantly affect the prediction of Nusselt number in both impinging and wall jet regions (fig. 5).

Evaluation of streamline

The streamlines resulting from flow impingement on a concave surface are shown in fig. 8. These lines are plotted for the case of D/B = 30, Re = 2960 and 4740, and h/B = 6 and 10. These figures indicate that after the impingement, the flow spreads on the concave surface in the

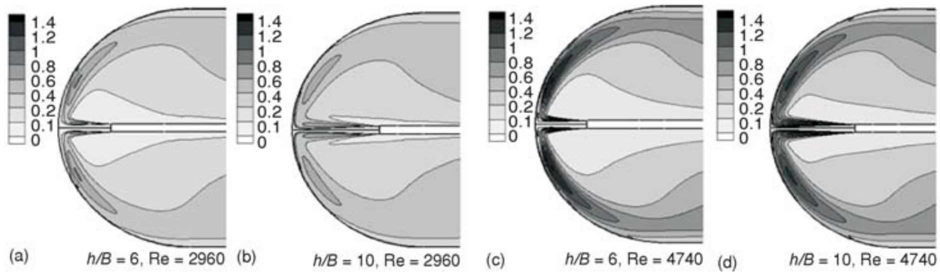

Figure 7. Contours of turbulence kinetic energy, *k*, along concave surface [m^2s^{-2}]

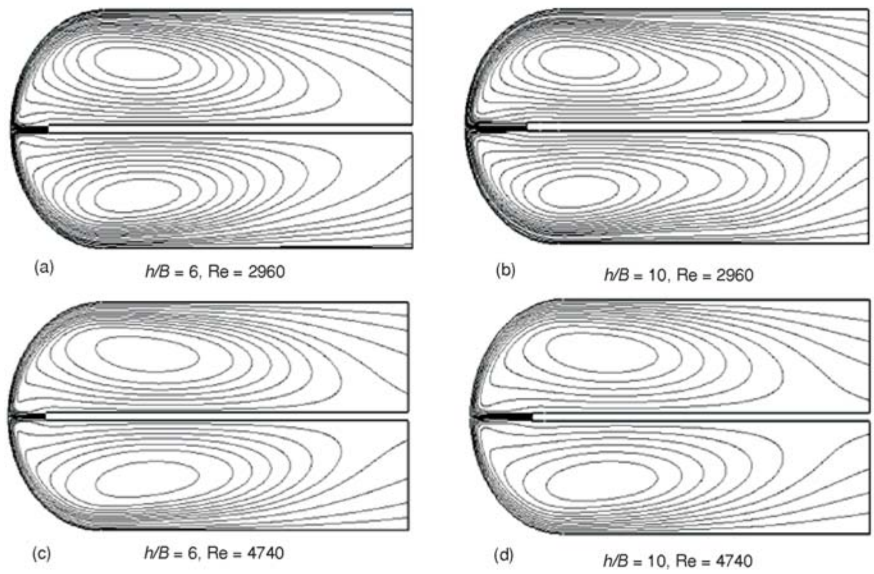

Figure 8. Streamline plots in the flow domain for different Reynolds number and various *h/B*

form of a wall jet in circumferential direction. The boundary layer thickness in the wall jet region increases with the increase of the distance between jet nozzle and target surface. It can be observed that with the reduction of the Reynolds number, the flow entrainment expands and becomes larger.

Effect of turbulent Prandtl number

Figure 9 shows the effects of turbulent Prandtl number (as the only prescribed constant affecting the modeling of turbulent heat transfer) on the distribution of local Nusselt number for the case of $D/B = 30$, which have been determined by using the LS-Yap model. According to eq. (7), higher values of Pr_t increase the heat diffusion along the wall and decrease temperature gradient in y-direction. Decreasing the turbulent Prandtl number improves the agreement of the local Nusselt number distribution compared to the experimental data.

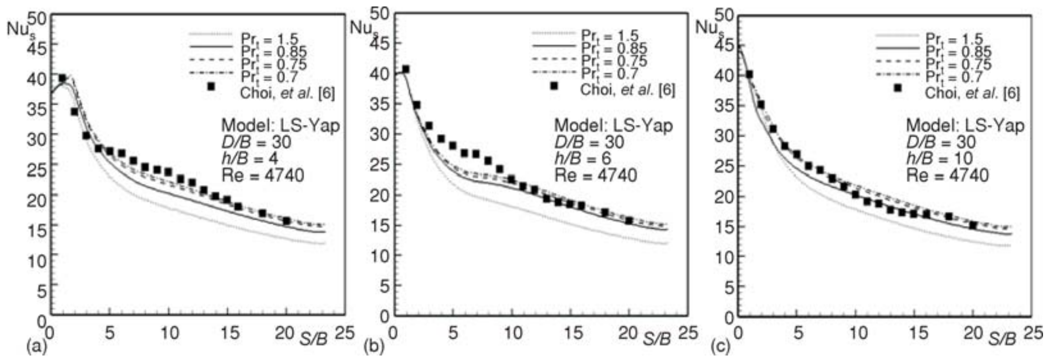

Figure 9. Effect of turbulent Prandtl variation on the Nusselt number

Conclusions

The flow and heat transfer resulting from impingement of a turbulent slot jet onto a semi-circular concave surface were numerically investigated. The obtained results were compared with the experimental data of Choi *et al.* [6]. Some of the conclusions reached in this study are outlined.

- The results presented by the low-Reynolds k-ε models indicate that these models are not capable to predict the Nusselt number distribution reasonably in the entire impingement surface length.
- Applying the Yap correction leads to lower prediction of velocity near the wall region. The reduction of predicted velocity, consequently improves the over-prediction of Nusselt number in impingement region.
- Applying the Yap correction to the low-Reynolds LS model predict the local Nusselt number in good agreement with a wide range of experimental results.
- The jet to target separation distance has no significant effect on the enhancement of the stagnation Nusselt number predicted by LS-Yap model.
- With the increase of the nozzle-to-surface distance and consequently the reduction of jet velocity, the numerical results show more agreement with the available experimental data.
- By decreasing the turbulent Prandtl number, the agreement of the local Nusselt number distribution is improved compared to the experimental data.

Nomenclature

B – slot width, [m]
c_p – specific heat, [Nmkg^{-1}K^{-1}]
D – diameter of the concave surface, [m]
h – nozzle-to-surface spacing, [m]
k – turbulent kinetic energy, [m2s^{-2}]
l – turbulence length scale, (= $k^{3/2}/\varepsilon$)
l_e – near wall equilibrium length scale, (= 2.495y)
Nu_s – Nusselt number, [= $q''(T_W - T_e)2B/\lambda$]
Pr_t – turbulent Prandtl number, eq. (7)
q'' – heat flux at the impingement surface, [Wm^{-2}]
Re – Reynolds number at the jet exit, (= $\rho U_e 2B/\mu$)
r – distance along the jet axis from the jet exit, [m]
S – arc distance along the concave surface, [m]
T_e – jet exit temperature, [K]
T_W – impingement surface temperature, [K]
U_i, U_j – time averaged velocity component in x- and y-directions, [ms^{-1}]
U_e – jet exit velocity, [ms^{-1}]
x_i, x_j – co-ordinates, [m]
y^+ – dimensionless distance between the wall and the first node

Greek symbols

ε – time rate of turbulent kinetic energy dissipation, (= $k^{3/2}/l$)

λ – thermal conductivity, [$Wm^{-1}K^{-1}$]
μ – dynamic viscosity
μ_t – turbulent or eddy viscosity, eq. (6), [$kgm^{-1}s^{-1}$]
r – fluid density, [kgm^{-3}]
ω – specific dissipation rate of turbulent kinetic energy, (= $k^{1/2}/l$)

References

[1] Martin, H., Heat and Mass Transfer between Impinging Gas Jets and Solid Surfaces, in: Advances in Heat Transfer (Eds. P. H. James, F. I. Thomas), Elsevier, 1977, pp. 1-60
[2] Mujumdar, A. S., Impingement Drying, in: *Hand Book of Industrial Drying*, 3rd ed. (Ed. A. S. Mujumdar), Taylor & Francis Group, New York, USA, 2007, pp. 385-395
[3] Sharif, M. A. R., Mothe, K. K., Evaluation of Turbulence Models in the Prediction of Heat Transfer Due to Slot Jet Impingement on Plane and Concave Surfaces, *Numerical Heat Transfer, Part B: Fundamentals, 55* (2009), 4, pp. 273-294
[4] Sharif, M. A. R., Mothe, K. K., Parametric Study of Turbulent Slot-Jet Impingement Heat Transfer from Concave Cylindrical Surfaces, *International Journal of Thermal Sciences, 49* (2010), 2, pp. 428-442
[5] Fregeau, M., Saeed, F., Paraschivoiu, I., Numerical Heat Transfer Correlation for Array of Hot-Air Jets Impinging on 3-Dimensional Concave Surface, *Journal of Aircraft, 42* (2005), 3, pp. 665-670
[6] Choi, M., *et al*., Measurements of Impinging Jet Flow and Heat Transfer on a Semi-Circular Concave Surface, *International Journal of Heat and Mass Transfer, 43* (2000), 10, pp. 1811-1822
[7] Eren, H., *et al*., Nonlinear Flow and Heat Transfer Dynamics of a Slot Jet Impinging on a Slightly Curved Concave Surface, *International Communications in Heat and Mass Transfer, 33* (2006), 3, pp. 364-371
[8] Hrycak, P., Heat Transfer from a Row of Impinging Jets to Concave Cylindrical Surfaces, *International Journal of Heat and Mass Transfer, 24* (1981), 3, pp. 407-419
[9] Kayansayan, N., Kucuka, S., Impingement Cooling of a Semi-Cylindrical Concave Channel by Confined Slot-Air-Jet, *Experimental Thermal and Fluid Science, 25* (2001), 6, pp. 383-396
[10] Lee, D. H., *et al.*, Technical Note the Effect of Concave Surface Curvature on Heat Transfer from a Fully Developed Round Impinging Jet, *International Journal of Heat and Mass Transfer, 42* (1999), 13, pp. 2489-2497
[11] Hosseinalipour, S. M., Mujumdar, A. S., Comparative Evaluation of Different Turbulence Models for Confined Impinging and Opposing Jet Flows, *Numerical Heat Transfer, Part A: Applications, 28* (1995), 6, pp. 647-666
[12] Seyedein, S. H., *et al*., Modelling of a Single Confined Turbulent Slot Jet Impingement Using Various *k-ε* Turbulence Models, *Applied Mathematical Modelling, 18* (1994), 10, pp. 526-537
[13] Souris, N., Liakos, H., Founti, M., Impinging Jet Cooling on Concave Surfaces, *AIChE Journal, 50* (2004), 8, pp. 1672-1683
[14] Wang, S. J., Mujumdar, A. S., A Comparative Study of Five Low Reynolds Number *k-ε* Models for Impingement Heat Transfer, *Applied Thermal Engineering, 25* (2005), 1, pp. 31-44
[15] Kumar, B. V. N. R., Prasad, B. V. S. S. S., Computational Flow and Heat Transfer of a Row of Circular Jets Impinging on a Concave Surface, *Heat Mass Transfer, 44* (2008), 6, pp. 667-678
[16] Yang, Y.-T., *et al*., Numerical Study of Turbulent Slot Jet Impingement Cooling on a Semi-Circular Concave Surface, *International Journal of Heat and Mass Transfer, 54* (2011), 1-3, pp. 482-489
[17] Bazdidi-Tehrani, F., *et al*., Normal-Velocity Relaxation and Higher Order Algebraic Heat Flux Models Applicable to an Axisymmetric Impinging Jet, *Journal of Dispersion Science and Technology, 33* (2011), 4, pp. 556-564
[18] Behera, R. C., *et al*., Numerical Study of Interrupted Impinging Jets for Cooling of Electronics, Components and Packaging Technologies, *IEEE Transactions on, 30* (2007), 2, pp. 275-284
[19] Abid, R., Evaluation of Two-Equation Turbulence Models for Predicting Transitional Flows, *International Journal of Engineering Science, 31* (1993), 6, pp. 831-840
[20] Lam, C. K. G., Bremhorst, K., A Modified Form of the *k-ε* Model for Predicting Wall Turbulence, *Journal of Fluids Engineering, 103* (1981), 3, pp. 456-460
[21] Launder, B. E., Sharma, B. I., Application of the Energy-Dissipation Model of Turbulence to the Calculation of Flow near a Spinning Disc, *Letters in Heat and Mass Transfer, 1* (1974), 2, pp. 131-137
[22] Abe, K., *et al.*, A New Turbulence Model for Predicting Fluid Flow and Heat Transfer in Separating and Reattaching Flows – II. Thermal Field Calculations, *International Journal of Heat and Mass Transfer, 38* (1995), 8, pp. 1467-1481

[23] Chang, K. C., *et al*., A Modified Low-Reynolds-Number Turbulence Model Applicable to Recirculating Flow in Pipe Expansion, *J Fluids Eng, 117* (1995), 3, pp. 417-423
[24] Hsieh, W. D., Chang, K. C., Calculation of Wall Heat Transfer in Pipeexpansion Turbulent Flows, *International Journal of Heat and Mass Transfer, 39* (1996), 18, pp. 3813-3822
[25] Daly, B. J., Harlow, F. H., Transport Equations in Turbulence, *Physics of Fluids, 13* (1970), 11, pp. 2634-2649
[26] Patankar, S. V., *Numerical Heat Transfer and Fluid Flow*, Hemisphere Publishing Company, Washington, USA, 1980
[27] Launder, B. E., Current Capabilities for Modelling Turbulence in Industrial Flows, *Applied Scientific Research, 48* (1991), 3, pp. 247-269
[28] Yap, C. R., Turbulent Heat and Momentum Transfer in Recirculating and Impinging Flows, Ph. D. thesis, University of Manchester, Institute of Science Technology, Manchester, UK, 1987.
[29] Behnia, M., *et al*., Prediction of Heat Transfer in an Axisymmetric Turbulent Jet Impinging on a Flat Plate, *International Journal of Heat and Mass Transfer, 41* (1998), 12, pp. 1845-1855

Paper submitted: October 3, 2012
Paper revised: May 17, 2013
Paper accepted: May 18, 2013