APPLICATION OF SECOND MOMENT CLOSURE AND HIGHER ORDER GENERALIZED GRADIENT DIFFUSION HYPOTHESIS TO IMPINGEMENT HEAT TRANSFER

Farzad Bazdidi-Tehrani and Mehran Rajabi-Zargarabadi
Department of Mechanical Engineering, Iran University of Science and Technology, Tehran 16846-13114, Iran
contact: bazdid@iust.ac.ir

Received October 2007, Accepted February 2008
No. 07-CSME-41, E.I.C. Accession 3010

ABSTRACT

This paper discusses the importance of turbulent heat flux modeling in predicting an impinging flow. A higher order version of the generalized gradient diffusion hypothesis (HOGGDH) is employed for the simulation of turbulent heat flux in impingement heat transfer. The flow field is modeled with both high and low Reynolds second moment closure turbulence models. For the high Reynolds second moment closure both GGDH and HOGGDH are not capable of capturing the shape of local Nusselt number profile in the impingement region. Combination of the low Reynolds second moment closure with either GGDH or HOGGDH models can reasonably predict the local Nusselt number distribution in comparison with the available experimental data. Results show that the HOGGDH over-predicts the turbulent heat transfer and the local Nusselt number particularly in the impingement zone.

APPLICATION DE DEUXIÈME FERMETURE DE MOMENT ET DE PLUS HAUT ORDRE À GÉNÉRALISER L'HYPOTHÈSE DE DIFFUSION DE GRADIENT AU TRANSFERT DE CHALEUR D'IMPACT

RÉSUMÉ

Ce papier exprime l'importance de modélisation du flux de chaleur turbulent pour prédire d'un écoulement d'impact. Une plus haute version d'ordre de l'hypothèse de diffusion de gradient généralisée (HOGGDH) est employée pour la simulation de flux de chaleur turbulent dans le transfert de chaleur d'impact. Le champ d'écoulement est étudié avec l'haute et le bas nombre de Reynolds sur le modèle de deuxièmes fermentures de moment turbulent. Pour l'haute Reynolds D.F.M. au cum des deux modèles (GGDH et HOGGDH) ne capturent pas le nombre local de Nusselt dans la région d'impact. La combinaison du bas nombre de Reynolds D.F.M. avec les modèles de GGDH ou HOGGDH peuvent prédire raisonnablement le nombre de Nusselt local en comparaison des données expérimentales disponible. Les résultats moment que le HOGGDH pas dessus prédit le transfert de chaleur turbulente et le nombre de Nusselt local Particulièrement dans la zone d'impact.
INTRODUCTION

Jet impingement cooling is widely used in many applications where high convective heat transfer rates are required. These applications include the drying of textiles and films and the cooling of gas turbine blades and combustor walls. The flow and heat transfer of an impinging jet depend on many parameters such as the non-dimensional nozzle-to-plate spacing, Reynolds number, Prandtl number and the non-dimensional distance from the stagnation point. A number of heat transfer studies have investigated these parameters in jet impingement on flat surfaces both numerically and experimentally [1-3].

Many numerical studies have focused on the effect of turbulence models on the accuracy of predicting the flow and heat transfer in impingement cooling. Craft et al. [4] have shown that Reynolds averaged turbulence models are not able to yield satisfactory results for an impinging jet. Behnia et al. [5] have studied numerically the jet impingement at high Reynolds number flows using various turbulence models. They have mentioned the importance of near-wall modeling.

Plat et al. [6] have compared experimental and numerical results for impingement heat transfer. They have shown that the standard $k-\varepsilon$ model with various wall functions fails to predict the stagnation heat transfer accurately. They have recommended the low Reynolds number $k-\varepsilon$ models as well as some advanced turbulence models to be tested for jet impingement. Shi et al. [7] have systematically studied the effects of turbulence models, near wall treatments, turbulence intensity, jet Reynolds number and boundary conditions on the heat transfer under a turbulent slot using the standard $k-\varepsilon$ and second moment closure models. Their results have indicated that both the standard $k-\varepsilon$ and second moment closure models predict the heat transfer rates inadequately, especially for low aspect ratios. They have recommended that for wall-bounded flows, it is necessary to integrate the equations through the viscous sublayer using finer grids with the low Reynolds turbulence models.

Wang and Mujumdar [8] have performed a comparative evaluation of five low Reynolds number $k-\varepsilon$ models for impingement heat transfer. They have proposed the Yap correction for reducing the turbulence length scale in the near-wall region with low Reynolds number $k-\varepsilon$ models. They have found that for most of the models, the Yap correction is capable of improving the predicted local Nusselt number showing good agreement with the experimental data in both stagnation and wall jet regions.

Sunden et al. [9] have validated the recent advances in computational methods for the simulation of impingement and forced convection. They have demonstrated that the linear and non-linear low Reynolds two-equation turbulence models such as $\nu^2 - f$ and $k-\omega$ can be used for impinging jet heat transfer predictions with reasonable success. Beaubert and Viazzo [10] have computed the flow field of plane impinging jets at moderate Reynolds numbers using the large eddy simulation model. Their results have illustrated favorable agreement with the available experimental data. They have also concluded that the effect of jet Reynolds number between 3000 and 7500 is significant both on the near and far field structures.

In the above-mentioned works and almost all of the numerical studies in impinging flow, several models and modifications have been evaluated for the better prediction of Reynolds stress tensor in the averaged momentum equations. For the thermal field computation, the generalized gradient diffusion hypothesis (GGDH) of Daly and Harlow [11] has been applied to describe the turbulent heat flux vector. However, it is well known that it can not predict the streamwise heat flux component reasonably well [12]. It is hence important to apply a more accurate model for predicting the turbulent heat flux vector which has a significant effect on the near-wall heat transfer rate [13-17].

Several models have been proposed for turbulent heat flux having better accuracies for the streamwise component [13, 14]. Amongst these, the model of Suga and Abe [15] has a generally expanded form of the GGDH. This is introduced as the higher order GGDH (HOGGDH) model which is formulated in
combination with the non-linear eddy viscosity model. Since it is reasonably expected that coupling it with a second moment closure model is more suitable for predicting complex thermal fields [17]. The higher order GGDH model has been validated for several turbulent heat transfer applications such as curved ducts [16, 17] and obstacle flow [18]. Results have shown that the HOGGDH heat flux model generally improves the predictions of the standard GGDH model due to a reasonable prediction of the streamwise heat flux component near the wall. Hence, it is important to accurately predict the turbulent heat flux for complex flows.

The aim of the present paper is to investigate the effect of turbulent heat flux modeling on the prediction of the heat transfer in impingement cooling. Hence, the second moment closure model in combination with the higher order GGDH model is employed for simulating the turbulent flow and heat transfer in a single slot jet. The results are compared with the simple GGDH model and the available experimental data.

PROBLEM STATEMENT

The two-dimensional flow geometry and the boundary conditions related to a slot jet are illustrated in Fig. (1). Due to the symmetry, only a half of the domain is considered for the numerical simulation.

The boundary conditions employed in the present work are identical to the experimental study of Van Heiningen [19]. The impinging surface is specified as an isothermal wall and the top confinement wall is set to have a constant temperature equal to that of the jet. Uniform temperature and 1/7th power law velocity profile are assumed at the jet nozzle inlet [8]. Symmetry and outflow boundary conditions are assumed at the symmetry and outlet planes. A no-slip condition is applied at the confinement walls. The turbulence intensity and length scale at the nozzle inlet are set as 2% and 0.07W, respectively [8].

GOVERNING EQUATIONS AND TURBULENCE MODELS

The governing transport equations are the continuity, momentum and energy equations, as represented by equations (1) - (3). The Reynolds stress transport equations and the turbulence dissipation rate are added through the turbulence model.

\[
\frac{\partial (\rho)}{\partial t} + \frac{\partial (\rho U_i)}{\partial x_i} = 0
\]

\[
\frac{\partial (\rho U_i)}{\partial t} + \frac{\partial (\rho U_i U_j)}{\partial x_j} = \frac{\partial p}{\partial x_i} + \frac{\partial }{\partial x_i} \left( \mu \frac{\partial \theta}{\partial x_i} - \rho u_j u_j \right)
\]

\[
\frac{\partial (\rho \theta)}{\partial t} + \frac{\partial (\rho U_i \theta)}{\partial x_i} = \frac{\partial }{\partial x_i} \left( \mu \frac{\partial \theta}{\partial x_i} - \rho u_j \theta \right)
\]
where, $u_i u_j$ and $u_i \theta$ are known as the Reynolds stress tensor and the turbulent heat flux vector, respectively. Both of these terms need to be modeled.

**FLOW FIELD**

**Second Moment Closures**

The transport equation for the Reynolds stress, $u_i u_j$, takes the following form [20]:

$$\frac{Du_i u_j}{Dt} = P_y + D_{y, L} + D_{y, T} + \phi_y - \varepsilon_y$$

where, the molecular diffusion, $D_{y, L}$, and the stress production, $P_y$, do not require any modeling:

$$D_{y, L} = \frac{\partial}{\partial x_i} \left( \nu \frac{\partial u_i u_j}{\partial x_j} \right)$$

$$P_y = -\left( u_i u_j \frac{\partial U_j}{\partial x_i} + u_j u_i \frac{\partial U_i}{\partial x_j} \right)$$

The remaining terms including the turbulent diffusion, $D_{y, T}$, the pressure strain, $\phi_y$, and the dissipation, $\varepsilon_y$, need to be modeled so as to close the equations. The assumptions required to close the set of equations are performed from the model of Hanjalic and Launder [20]. The turbulent diffusion, $D_{y, T}$, is modeled as in equation (7).

$$D_{y, T} = \frac{\partial}{\partial x_k} \left[ C_K k \left( u_i u_j \frac{\partial u_k}{\partial x_i} + u_k u_j \frac{\partial u_i}{\partial x_k} + u_k u_i \frac{\partial u_j}{\partial x_k} \right) \right], C_K = 0.11$$

The classical approach to modeling $\phi_y$ employs the following decomposition:

$$\phi_y = \phi_{y1} + \phi_{y2} + \phi_{y,w}$$

where, $\phi_{y1}$ is the slow pressure-strain term, $\phi_{y2}$ is the rapid pressure-strain term, and $\phi_{y,w}$ is the wall-reflection term. The models of the $\phi_{y1}$ and $\phi_{y2}$ including their coefficients are listed as follows. Full details are given by Hanjalic and Launder [20].
\[ \phi_{y1} = -C_1 \frac{\varepsilon}{k} \left( u_i u_j - 2 \delta_{ij} / 3k \right), \quad C_1 = 2.8 \]

\[ \phi_{y2} = a_{ij}^m \left( \frac{\partial U_m}{\partial x_i} + \frac{\partial U_i}{\partial x_m} \right) \]

**Near-Wall Modifications**

The set of Reynolds stress transport equations is originally regarded as a high Reynolds number turbulence model and is used outside the viscous affected layer close to a wall. The viscous affected layer can be included by incorporating wall damping functions and modifications of the \( \varepsilon \)-transport equation, the anisotropic dissipation, \( \varepsilon_y \), and the wall reflection pressure-strain term, \( \phi_{y,w} \) [21]. In the present study, all of these modifications are carried out by employing the models of Lai and So [22]. In general, the near-wall \( \varepsilon \)-transport equation may be written in the following form:

\[ \frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_i} \left[ C_\varepsilon \frac{k}{\varepsilon} \frac{\partial \varepsilon}{\partial x_j} + \nu \frac{\partial \varepsilon}{\partial x_i} \right] + C_{\varepsilon 1} \frac{\varepsilon}{k} \left( -P + \Psi - C_{\varepsilon 2} f_{\varepsilon} \frac{\varepsilon}{k} + \zeta \right) \]

where, \( \Psi \) denotes the additional production term, \( \zeta \) is the near-wall modification term and \( f_{\varepsilon} \) is the near-wall damping function of \( \varepsilon \)-transport equation. Table (1) shows the modification of \( \varepsilon \)-transport equation, near-wall modifications for \( \varepsilon_y \) and \( \phi_{y,w} \), proposed by Lai and So [22].

**Table 1. Near-wall modifications of Lai and So [22]**

<table>
<thead>
<tr>
<th>( \varepsilon )-Transport equation</th>
<th>Anisotropic dissipation, ( \varepsilon_y )</th>
<th>Wall reflection, ( \phi_{y,w} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \Psi = C_{\varepsilon 1} \frac{\varepsilon}{k} f_{\varepsilon} ) ( f_{\varepsilon} ) ( P ) ( \sigma )</td>
<td>( \varepsilon_y = \frac{2}{3} \varepsilon (1 - f_{\varepsilon}) \varepsilon_y + \frac{\nu}{\partial x_i} )</td>
<td>( \phi_{y,w} = f_{\varepsilon} \left( C_1 \varepsilon \frac{\nu u_j}{k} - 2 \delta_{ij} / 3k \right) )</td>
</tr>
<tr>
<td>( \sigma = 1 - 0.6 \exp \left[ - \frac{Re}{10^4} \right] )</td>
<td>( f_{\varepsilon} = \exp \left[ \frac{1}{2} \left( \frac{R_i}{64} \right) \right] )</td>
<td>( \phi_{y,w} = f_{\varepsilon} \left( C_1 \varepsilon \frac{\nu u_j}{k} - 2 \delta_{ij} / 3k \right) )</td>
</tr>
<tr>
<td>( \zeta = \left[ 1 + 3 \frac{u_i u_j}{k} \right] )</td>
<td>( f_{\varepsilon} = \exp \left[ \frac{1}{2} \left( \frac{R_i}{64} \right) \right] )</td>
<td>( \phi_{y,w} = f_{\varepsilon} \left( C_1 \varepsilon \frac{\nu u_j}{k} - 2 \delta_{ij} / 3k \right) )</td>
</tr>
<tr>
<td>( \varepsilon_y = \frac{\varepsilon}{\partial x_i} \left[ f_{\varepsilon} \left( \frac{\nu u_j}{k} + \frac{u_i u_j}{\partial x_i} \right) \right] )</td>
<td>( P = - (\nu u_j + u_i \frac{\partial U_m}{\partial x_i} + u_j \frac{\partial U_i}{\partial x_m}) )</td>
<td></td>
</tr>
<tr>
<td>( C_\varepsilon = 0.15 )</td>
<td>( = u_i \frac{\partial U_i}{\partial x_j} )</td>
<td></td>
</tr>
<tr>
<td>( C_{\varepsilon 1} = 1.35 )</td>
<td>( C_{\varepsilon 2} = 1.8 )</td>
<td></td>
</tr>
</tbody>
</table>

**TURBULENT HEAT FLUX MODELING**

The simplest way to model the turbulent heat flux, \( \overline{u_i \theta} \), is to employ the generalized gradient diffusion hypothesis (GGDH) model, with a prescribed turbulent Prandtl number, \( \text{Pr}_\theta \), which is commonly assumed as 0.85 for near-wall flows [11]:

\[ \overline{u_i \theta} = - \nu \frac{\partial \theta}{\partial x_i} / \text{Pr}_\theta \]
The GGDH model is far more successful when the Reynolds stresses $\overline{u_iu_j}$ are reasonably captured all the way to the wall by a flow field turbulence model. The model form may be written as [12]:

$$\overline{u_i\theta} = -c_\theta \tau \overline{u_iu_j} \frac{\partial \Theta}{\partial x_j}$$

The GGDH for turbulent heat flux still under-predicts the streamwise heat flux component. Hence, Suga and Abe [15] have extended the applicability of the GGDH, by introducing the higher-order (quadratic) terms and they have exactly reproduced each heat flux component. Their higher-order GGDH (HOGGDH) model may be written as follows.

$$\overline{u_i\theta} = -c_\theta k \tau (\sigma_{ij} + \alpha_{ij}) \frac{\partial \Theta}{\partial x_j}$$

where, the symmetric tensor contains linear and quadratic terms consisting of the Reynolds stress tensor as:

$$\sigma_{ij} = c_{\sigma_1} \overline{u_iu_j} + c_{\sigma_2} \overline{u_iu_j} / k^2$$

and the asymmetric part is modeled as:

$$\alpha_{ij} = c_{\alpha_1} \left( \frac{\Omega_{ij} \overline{u_iu_j}}{k} + \frac{\Omega_{ij} \overline{u_iu_j}}{k} \right)$$

Table (2) presents the details of model coefficients and functions for the higher-order GGDH [15].

<table>
<thead>
<tr>
<th>Table 2. Model coefficients and functions for HOGGDH [15]</th>
</tr>
</thead>
<tbody>
<tr>
<td>$c_{\sigma_1} = 0.2 - f_{\sigma_1} S \frac{0.02 \exp \left[ - \left( S / 2.2 \right)^2 \right]}{S + 0.2}$</td>
</tr>
<tr>
<td>$f_{\sigma_1} = \frac{1}{1 + \left( \frac{\sigma_{ij}}{0.08} \right)^2}$</td>
</tr>
<tr>
<td>$g_{\sigma} = 0.3 \left[ 1 - \exp \left( - \frac{R_{\sigma}}{70} \right) \right]$</td>
</tr>
</tbody>
</table>

**NUMERICAL SOLUTION**

The present two-dimensional computational domain consists of rectangular cells, which have been carefully generated and examined for grid independence solution. Fig. (2) illustrates two present strategies of grid generation for the low and high Reynolds second moment closure (SMC) turbulence models. For low Reynolds second moment closure formulation, it is important that the $y^+$ value of the grid points closest to the wall be of the order of unity. Here, as shown in Fig. (2), the grid has a strong clustering close to the walls to reach $y^+ \sim 1$ in the wall adjacent cells. For high Reynolds second moment
closure, the grid is generated uniformly in a way that the $y^+$ value of wall adjacent cells is kept in $y^+ \approx 30$. The viscous-affected region between the wall and the fully turbulent field is bridged by the standard wall function [23].

Fig. (2) Close up view of present grid generation, (a) Low Reynolds SMC and (b) High Reynolds SMC

Fig. (3) represents the effect of grid size on the predicted streamwise local Nusselt number distribution. The low Reynolds SMC and GGDH models are employed for this solution. All the present grid sizes are fine enough to reach $y^+ \approx 1$ in the wall adjacent cells. The grid size of 30,000 (400*75) cells provides an almost satisfactory solution for the low Reynolds SMC formulation. A finer grid size of 50,000 (500*100) cells causes no significant changes in the present results. For the high Reynolds SMC formulation, the grid size of 400*75 with a uniform distribution also satisfies the independence of present results from the grid size with the $y^+$ value of wall adjacent cells kept in $y^+ \approx 30$.

Fig. (3) Effect of grid size on the predicted streamwise $\text{Nu}_x$, (H/W=6, $Re_{jet} = 5200$)
In the present work, numerical computations have been carried out using an extended CFD code, called ISAAC [24]. ISAAC employs a second-order, upwind, Finite-Volume method where viscous terms are discretized by a second-order central difference scheme. Mean and turbulence equations are solved coupled using an implicit spatially split, diagonalized approximate factorization solver [25]. ISAAC has been extensively used by other researchers for various cases [26-28].

Multigrid acceleration is applied to the mean flow equations and mesh sequencing (full multigrid) is employed to provide an initial solution. The convergence criterion of the solution is assumed when the sum of the normalized residuals is reduced to about $1 \times 10^{-5}$.

**DISCUSSION OF RESULTS**

Two sets of aspect ratios and jet Reynolds numbers are selected from the available experimental data of Van Heiningen [19] as the test cases used for comparisons with the present numerical results concerning confined jet impingement flows. These are regarded as case (A): $H/W=6$, $Re_{jet}=5200$ and case (B): $H/W=2.6$, $Re_{jet}=10400$.

Fig. (4) depicts a comparison of the predicted streamlines within the confined jet flow geometry for cases (A) and (B). A recirculation zone is observed for both cases and it is identical with both the high and low Reynolds second moment closure turbulence models. For case (A) the recirculation zone is extended up to approximately an axial distance of $x/L=0.35$. The recirculation zone is stretched further up to $x/L=0.6$ for case (B), indicating the significance of a larger jet Reynolds number based on an increased slot width (i.e., $Re = \rho V_j W / \mu$).

![Fig. (4) Comparison of streamlines](image)

Fig. (5), in association with Fig. (4), shows the predicted contours of velocity vectors for cases (A) and (B). In the present study, the jet Reynolds number is increased by enhancing the slot width while the jet inlet velocities are almost identical. Thus, the most significant difference between the two cases lies in the jet-affected region. For case (A) a main vortex is formed in the confined channel. The jet is penetrated and then spread out in the vicinity of the target wall up to nearly $x/L=0.35$. For case (B) the jet is spread out further and a larger expanded vortex is formed up to $x/L=0.6$. The formation of flow recirculation is mainly due to the suction effect of the jet near the inlet region.
Fig. (5) Contours of velocity vectors, (a) H/W=6, Re_{jet} = 5200 and (b) H/W=2.6, Re_{jet} = 10400

Fig. (6) represents the variations of streamwise local Nusselt number, Nu_x, for case (A) applying two different heat flux models. A maximum near the stagnation point (x/W ≈ 0) and a subsequent reduction in Nu_x are shown by both the experimental data [19] and the present results based on the predictions of combined low Reynolds second moment closure (LRSMC) turbulence model and turbulent heat flux (GGDH and HOGGDH) models.

It can be seen that the shape of Nu_x profile is captured by both heat flux models quite well. A reasonable agreement with a difference of less than 20% on average exists between the present combined LRSMC - GGDH and the experimental results. An over-prediction of Nu_x by approximately 30% on average, particularly in the impingement zone (x/W<5), is observed when applying the HOGGDH model. This could be due to a significant increase in the estimation of turbulent heat flux, as discussed in Fig. (8). The results based on the predictions of high Reynolds SMC turbulence model give false capturing of Nu_x profile shape, particularly in the impingement zone. In addition, the difference between HOGGDH and
GGDH models, combined with high Reynolds SMC, is not significant due to applying a wall function in the viscous-affected region.

Fig. (7) illustrates the variations of streamwise $Nu_x$ for case (B). The shape of $Nu_x$ profile, characterized by two maxima at approximately $x/W<0.5$ and $x/W=7$, is fairly captured by the present predictions based on combined low Reynolds second moment closure (LRSMC) model and GGDH / HOGGDH models. Once again, a reasonable agreement with a difference of less than 20% on average is displayed between the present combined LRSMC - GGDH and the experimental results [19]. Similar to Fig. (6), an over-prediction of $Nu_x$ by almost 25% on average, particularly in the impingement zone, is shown when employing the HOGGDH model. The present results based on the high Reynolds SMC model are not in agreement with the experimental data at all and false predictions of $Nu_x$ distribution are observed. This means that applying a wall function fails to predict the viscous-affected region. Also, applying the HOGGDH model just causes the predicted $Nu_x$ to increase without affecting its profile shape.

According to Figures (6) and (7), the average local Nusselt number for case (B) is relatively greater than that for case (A) while the maximum values (in the stagnation point) are almost the same. This is due to the effect of a larger jet Reynolds number on the size and formation of a recirculation zone, as discussed previously in Figures (4) and (5).

![Fig. (7) Variations of streamwise local Nusselt number (H/W=2.6, Re_{jet} = 10400)](image)

Figures (8) and (9) show the present predictions of the total and lateral turbulent heat flux distributions, respectively. The low Reynolds SMC model is hereby applied and a comparison between two different turbulent heat flux models is made for cases (A) and (B). All of the profiles are plotted for a half of the channel height so as to capture the near wall gradients. For $y/H>0.5$, the velocity and temperature gradients are negligible. In the near-wall region ($y/H<0.1$), the HOGGDH model predicts considerably higher values than those of the GGDH model for both the total and lateral turbulent heat flux. Also, according to the $Nu_x$ results reported in Figures (6) and (7), it can be deduced that the HOGGDH model generally over-predicts the turbulent heat flux transfer. This is mainly due to the extreme dependence of this model on the velocity gradients and Reynolds stresses (see equations 14 and 15) which have large magnitudes in the near-wall region of an impinging flow.
Fig. (8) Turbulent heat flux distribution in the impinging flow
(a) H/W=6, Re_{jet} = 5200 and (b) H/W=2.6, Re_{jet} = 10400

Fig. (9) Lateral turbulent heat flux distribution in the impinging flow
(a) H/W=6, Re_{jet} = 5200 and (b) H/W=2.6, Re_{jet} = 10400

Fig. (10) represents the non-dimensional mean temperature distributions for cases (A) and (B), obtained by applying the two turbulent heat flux models. At small x/L, the temperature profiles predicted by the HOGGDH model are greater than those of the GGDH model, particularly in the near-wall region. This is in accordance with the higher turbulent heat flux values predicted by the HOGGDH model, as reported in...
Figures (8) and (9). As x/L increases, the turbulent mixing causes the temperature profiles to become more identical.

CONCLUSIONS

A numerical simulation of the turbulent heat transfer in a confined impinging jet flow has been performed. The effects of near-wall second moment closure modification and the higher-order GGDH model for the turbulent heat flux are evaluated. The main conclusions are as follows:

1- Turbulent heat flux model plays an important role in accurately predicting complex flows such as those encountered in the impingement jet flow. Application of more accurate models than the GGDH is recommended for wall-bounded heat transfer flows.

2- The high Reynolds second moment closure turbulence model combined with either HOOGGDH or GGDH turbulent heat flux model is unable to predict the local Nusselt number distribution due to incorporating the standard wall function into the viscous-affected region.

3- The low Reynolds modification of SMC turbulence model combined with either HOGGDH or GGDH provides an improvement in the predictions of the local Nusselt number profiles being in reasonable agreement with the experimental data.

4- The HOOGGDH model over-predicts the turbulent heat flux due to the extreme dependence of this model on the velocity gradients and Reynolds stresses. This results in an over-estimation of the local Nusselt number distribution particularly in the impingement zone.

5- Mean temperature profiles of the HOOGGDH model are significantly higher than those of the GGDH model at small x/L. As x/L increases, the turbulent mixing causes the profiles tend to be identical.
REFERENCES


NOMENCLATURE

H nozzle-to-plate spacing
h heat transfer coefficient
I turbulence intensity
k turbulent kinetic energy
k thermal conductivity
l turbulence length scale
L duct length
Nu Nusselt number, \( \Nu = \frac{h W}{k} \)
p static pressure
Re\textsubscript{jet} jet Reynolds number based on jet inlet velocity and slot width, \( \Re_{\text{jet}} = \frac{\rho V_j W}{\mu} \)
Re turbulent Reynolds number, \( Re = \frac{k^3}{\nu \varepsilon} \)
S\textsubscript{ij} strain tensor \( (\partial U_i / \partial x_j + \partial U_j / \partial x_i) \)
u\textsubscript{ij}, \( U_i \) instantaneous and mean velocity components
\( \overline{U} \) axial average velocity
\( u_{\mu ij} \) Reynolds stress tensor
\( \overline{u_i \theta} \) turbulent heat flux vector
\( V_j \) jet inlet velocity component in y direction
W slot width
\( y^* \) dimensionless distance, \( y^* = u \sqrt{\rho \mu} / \mu \)

Greek Symbols

\( \mu, \mu_l \) laminar and eddy viscosities
\( \nu, \nu_l \) laminar and eddy kinematic viscosities
\( \rho \) density
\( \varepsilon \) dissipation rate of turbulent kinetic energy
\( \varepsilon \)  dissipation rate of Reynolds stress
\( \bar{\varepsilon} = 2\nu(\partial \sqrt{k}/\partial x_i)(\partial \sqrt{k}/\partial x_j) \)
\( \varepsilon^* = 2\nu k/y^2; \)
\( \Theta \) mean temperature
\( \Theta^* \) non-dimensional temperature, \( \Theta^* = (\Theta_{imp} - \Theta)/(\Theta_{imp} - \Theta_{jet}) \)
\( \tau \) turbulent time scale
\( \Omega_y \) vorticity tensor \((\partial U_j/\partial x_i - \partial U_i/\partial x_j)\)