Numerical Simulations of Flow and Heat Transfer over Rib-Roughened Surfaces

A. Keshmiri*, M.A. Cotton and Y. Addad

School of Mechanical, Aerospace and Civil Engineering, The University of Manchester, Manchester M60 1QD, UK.

* e-mail: A.Keshmiri@postgrad.manchester.ac.uk

ABSTRACT

Flow and heat transfer are simulated numerically in a 2-dimensional rib-roughened duct for different values of the rib pitch-to-height ratio \((P/k)\). Three \(P/k\) ratios of 6, 9 and 12 are examined. The blockage ratio of these rectangular obstacles is 10% and the Reynolds number, based on the channel bulk velocity and hydraulic diameter, is fixed at 30,000. Two refined 'low-Reynolds-number' turbulence models, namely the Lien-Chen-Leschziner \(k-\varepsilon\) model and a variant of Durbin \(\nu^2f\) formulation are employed and the results were examined against experimental data. All computations were undertaken using the commercial CFD package ‘STAR-CD’. It was found that the \(\nu^2f\) model returned more accurate results than the \(k-\varepsilon\) closure. It was further found that the ratio \(P/k = 9\) produced the maximum heat transfer level and pressure loss.

1. INTRODUCTION

Rough surfaces are widely used to enhance convective heat transfer by the promotion of higher turbulence levels. The drawback to such roughening is an increase in frictional and form drag and consequently much effort has been devoted to the optimization of roughness designs.

There have been numerous studies of the flow dynamics and heat transfer characteristics of various rough surfaces. However, a full understanding of the detached flow physics remains elusive. This is largely due to experimental difficulties and high turbulence intensities that render measurement techniques inaccurate [1].

Numerical studies are becoming more common; the most widespread techniques adopted are based on solution of the Reynolds-Averaged Navier-Stokes (RANS) equations. In this approach the Reynolds stresses are computed using a supplementary turbulence model. The choice of turbulence model plays a critical role in determining the accuracy of the simulations. In the present study, two turbulence models of the 'low-Reynolds-number' type are employed. (A number of models of this general class have been used by the authors and their colleagues in the computation of smooth surface buoyancy-affected, or 'mixed convection' flows [2-4].)

Depending on the surface configuration, one can divide the roughness into two different categories. The first category corresponds to a geometry where the gap between the two ribs is small (see Figure 1) and is occupied by a re-circulating flow, as shown in Figure 2 (a). This type of roughness is known as ‘\(d\)-type’ roughness. The second type refers to a situation where the gap between the ribs is larger. This type of roughness is characterized by eddies that form behind a roughness element. This type of roughness is known as ‘\(k\)-type” roughness (see Figure 2 (b)).

The distinction between \(d\)- and \(k\)-roughness was first made by Perry et al. [5]. They observed that in several boundary layers over plates roughened by narrow spanwise square grooves, the effective roughness \(k\), was not proportional to the roughness height \((k)\), but to the boundary-layer thickness \((d)\), and thus this latter type of roughness was named \(d\)-type. On the other hand, they found that for some
rough surfaces where the gap between the grooves was wider, roughness effects showed clear dependence on the roughness height, hence $k$-type roughness.

Figure 2 – Regimes of the mean flow over rod roughness

Apart from the significance of the horizontal gap between the ribs, relative roughness height is also important. In [6], it is estimated that the channel height, $H$, should be 40-80$k$ in order to eliminate the direct effect of the roughness elements on the outer flow, otherwise it is likely that direct roughness effects would be felt across the channel height and thus the turbulent flow would no longer be categorized as flow over roughness elements, but rather as flow over surface-mounted bluff bodies.

There have been many experimental studies of the rib-roughened surfaces including those of Han et al. [7], Hirota et al. [8], Okamoto et al. [9], Rau et al. [10], Liou et al. [11] and, more recently, the works of Krogstad et al. [12] and Lee et al. [13].

In the present work, the simulations are compared with the data of Rau et al. [10]. In that study, detailed flow and heat transfer measurements were made in a square channel with ribs presenting a significant blockage ratio ($k/H = 0.1$). The results of the local measurements were discussed for three different $P/k$ ratios (6, 9 and 12) in a one-side-ribbed channel. Measurements for a two-side-ribbed channel were reported for $P/k = 9$.

Additionally, there also have been numerous attempts to model straight and inclined ribs in stationary and rotating passages and U-bends using RANS.

Archarya et al. [14] applied both linear and non-linear $k$-$\varepsilon$ models to successive two-dimensional rectangular ribs and found that the performance of the two models was similar, except that the non-linear model produced more realistic Reynolds stress distributions than the linear form in the region immediately above the ribs.

Liou et al. [15] studied ribbed surfaces numerically and experimentally, using the $k$-$\varepsilon$-$A$ algebraic stress and heat flux model and LDV measurements. They found good agreement between modelling and experimentation for a 2D case.

Iacovides and Raisee [16] examined the capabilities of low-Reynolds-number versions of Launder & Sharma $k$-$\varepsilon$ model [17] in predicting convective heat transfer in ribbed annular channels, pipes and plane channels. They obtained a more realistic heat transfer variation in the separation region and reasonable Nusselt number levels by employing a differential form of the Yap term in the $\varepsilon$-equation (a correction term for the near-wall turbulent length-scales which is independent of wall distance) [18].

Ooi et al. [19] carried out simulations of the flow and heat transfer in 3-dimensional rib-roughened ducts using the $\nu^2f$ and Spalart-Allmaras (S-A) turbulence models and compared their results with the experimental data of Rau et al. [10] and the $k$-$\varepsilon$ simulations of Chen and Patel [20]. Configurations with various geometrical parameters including pitch, rib height, and cavity depth were considered. They showed that while $k$-$\varepsilon$ model severely underestimates heat transfer levels, while the S-A model gave heat transfer results that were closer to the experimental data, but nonetheless the computed values of $Nu$ were still far from the measured values. They also found that heat transfer results generated by the $\nu^2f$ model were closest to the experimental values. However, none of the above models could capture the secondary flow structure which consequently led to incorrect predictions of $Nu$ on the heated side wall.

Iaccarino et al. [21] presented a detailed analysis of the capabilities of the $\nu^2f$ model in predicting heat transfer in rib-enhanced passages. They showed that the computed averaged values of Nusselt number closely matched the experimental data. However, local values of Nusselt number very close to the ribs were strongly affected by the rib thermal boundary condition. Their numerical data showed that heat transfer is dominated by convection upstream, and conduction downstream, of the rib.

2. Computational Framework
2.1 Theoretical Background

The mean flow equations are written in the ‘thin shear’, or ‘boundary layer’, approximation. Thus, in Cartesian tensor notation, the mean flow governing equations are as follows:
Continuity:
\[ \frac{\partial U_j}{\partial x_j} = 0 \]  

Momentum:
\[ \rho \frac{DU_j}{Dt} = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \frac{\mu + \mu_t}{\sigma_t} \frac{\partial U_j}{\partial x_j} \right) \]  

Energy:
\[ \rho \frac{DT}{Dt} = \frac{\partial}{\partial x_j} \left[ \frac{\mu + \mu_t}{\sigma_t} \frac{\partial T}{\partial x_j} \right] \]  

The turbulent Prandtl number is set to a constant value, \( \sigma_t = 0.9 \).

2.2 Numerical Details

The commercial CFD code, STAR-CD [22] was used to generate the results using the low-Reynolds-number Lien, Chen and Leschziner (LCL) \( k-\varepsilon \) model and a variant of Durbin \( v^2f \) model (hereafter, will be called Durbin \( v^2f \) model), the details of which can be found in Section 2.4. STAR-CD has been validated against in-house and industrial codes in earlier works by the Manchester group [2, 3].

All fluid properties are assumed to be constant. The momentum and turbulence equations are discretized using first-order upwind differencing and second-order central differencing schemes, respectively. The energy equation is discretized using ‘Monotone Advection and Reconstruction Scheme’ (MARS) [22]. The SIMPLE algorithm is adopted for pressure-velocity correction.

2.3 Geometry and Grid

The configuration considered here consists of a 2-dimensional channel with the lower wall roughened by square ribs of height, \( k \). As shown in Figure 3 the computational domain is of length 2\( P \), i.e. it includes 2 complete ribs. In all the simulations, the flow is assumed to be periodic and therefore cyclic boundary conditions are applied in the streamwise direction. The thermal boundary condition at the lower wall is one of uniform wall heat flux, whereas the upper wall is adiabatic.

Pitch-to-rib height ratios of 6, 9 and 12 are simulated. The hydraulic diameter in all cases is \( D_h = 0.1 \) m. The rib-to-channel hydraulic diameter ratio is \( k/D_h = 0.05 \) and the blockage ratio (\( k/H \)) is 10\%. The Reynolds number based on hydraulic diameter, \( Re \), is fixed at 30,000. The Prandtl number is set to 0.71 in all the calculations.

A number of carefully-conducted sensitivity tests have been applied to ensure the numerical reliability of the results presented below. Three structured Cartesian grids with 133,000 (\( P/k = 6 \)), 161,000 (\( P/k = 9 \)) and 189,000 (\( P/k = 12 \)) cells were used. Since low-Reynolds-number turbulence models are employed, the grids were generated so as to be very fine near the wall (the wall-adjacent cell extends only to \( y^+ \leq 0.5 \)).

2.4 Turbulence Models

The details of the two turbulence models used in the present study are given below.

Lien, Chen and Leschziner (LCL) \( k-\varepsilon \) model

The low-Reynolds-number \( k-\varepsilon \) model of Lien, Chen and Leschziner [23], is termed the ‘Standard Low-Reynolds-Number Model’ in the STAR-CD documentation [22]. The formulation is similar to the Launder-Sharma model [17], however, the \( \varepsilon \)-equation is cast in a slightly different form, in part to improve the convergence properties of the model.

Eddy viscosity in the LS model is obtained as:
\[ \mu_t = f_\mu C_\mu \rho k^2 / \varepsilon \]  
where
\[ f_\mu = \left[ (1 - e^{-0.0198 \cdot Re}) \right] \left[ 1 + \left( 5.29 / Re \right) \right] \]  

The transport equations for the turbulent kinetic energy, \( k \) and its dissipation rate, \( \varepsilon \), are given as:
\[ \rho \frac{Dk}{Dt} = \rho P_k + \frac{\partial}{\partial x_j} \left[ \frac{\mu + \mu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \right] - \rho \varepsilon \]
The coefficients of the LCL $k$-$\varepsilon$ model are quoted in Table 1.

### Table 1 - Coefficients of the LCL $k$-$\varepsilon$ model

<table>
<thead>
<tr>
<th>$C_\mu$</th>
<th>$\alpha_1$</th>
<th>$\sigma_\varepsilon$</th>
<th>$C_{\varepsilon 1}$</th>
<th>$C_{\varepsilon 2}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.09</td>
<td>1.0</td>
<td>1.22</td>
<td>1.44</td>
<td>1.92</td>
</tr>
</tbody>
</table>

#### Durbin $v^2f$ model

The original $v^2f$ model of Durbin [24] took as its starting point, the two-equation $k$-$\varepsilon$ model of Launder & Sharma [17]. It was designed to handle near-wall effects in turbulent boundary layers and to accommodate non-local effects. In place of a conventional damping function, a third transport equation is included for $v^2$ ($v$ is the wall-normal component of the fluctuating velocity vector). $v^2$ is used in a revised definition of turbulent viscosity given by the following equations:

$$
\mu_t = \rho C_\mu \frac{v^2}{T_s}
$$

where

$$
T_s = \max \left[ \frac{k}{\epsilon}, \frac{C_{\varepsilon 1}}{\eta} \left( \frac{v}{\epsilon} \right)^{0.5} \right]
$$

In addition, an elliptic equation for $f_{22}$, the redistribution term in the $v^2f$ equation, is included to account for near-wall and non-local effects.

STAR-CD uses a variant of Durbin [25] version of the $v^2f$ model. The governing equations of this particular model are given in [26] as follows:

$$
\rho \frac{D\varepsilon}{Dt} = C_{\varepsilon 1} \rho \left( \frac{\mu + \mu_t}{\sigma_\varepsilon} \right) \frac{\partial k}{\partial x_j} - C_{\varepsilon 2} f_{21} \rho \frac{\varepsilon}{k}
$$

where:

$$
P' = 1.33 \left( 1 - 0.3 e^{-Re_\gamma^3} \right)
$$

$$
\left( \frac{\mu t}{\mu} \right) vD + \frac{2 k}{\mu_t} \frac{k}{\mu} \frac{\partial k}{\partial x_j} e^{-0.00375 Re_\gamma^2}
$$

$$
Re_\gamma = k^2 l (\nu v)
$$

$$
Re_\gamma = \frac{\gamma k}{\nu}
$$

$$
f_{21} = 1 - 0.3 e^{-Re_\gamma^2}
$$

The coefficients of the Durbin $v^2f$ model are given in Table 2.

### Table 2 - Coefficients of the Durbin $v^2f$ model

<table>
<thead>
<tr>
<th>$C_\mu$</th>
<th>$\alpha_1$</th>
<th>$\sigma_\varepsilon$</th>
<th>$C_{\varepsilon 1}$</th>
<th>$C_{\varepsilon 2}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.22</td>
<td>1.0</td>
<td>1.3</td>
<td>1.4</td>
<td>1.9</td>
</tr>
<tr>
<td>$C_{L1}$</td>
<td>$C_{L2}$</td>
<td>$C_{\varepsilon 1}$</td>
<td>$C_{\varepsilon 2}$</td>
<td></td>
</tr>
<tr>
<td>1.4</td>
<td>0.3</td>
<td>0.23</td>
<td>70.0</td>
<td>6.0</td>
</tr>
</tbody>
</table>

#### 3. RESULTS AND DISCUSSIONS

Heat transfer results are shown in Figure 4. The Nusselt number is defined here as:

$$
Nu = \frac{\dot{q} D_h}{\lambda (T_w - T_0)}
$$

As in [10], all Nusselt number distributions for the ribbed duct calculations are normalized by the value associated with a smooth passage (the Dittus-Boelter equation):

$$
Nu_0 = 0.023 Re^{0.8} Pr^{0.4}
$$
However, the present LCL k-ε formulation has not been tested previously in this type of configuration.

Figure 5 shows the normalized average Nusselt number and friction coefficient for all three P/k ratios. Following [10], the Nusselt number has been averaged over the gap between the two ribs, while the average effective friction coefficient is based on the pressure drop (Δp) (Eqn. (22)) over one complete pitch (see Figure 3 of [10] for more details).

\[
c_f = \frac{\tau_w}{0.5 \rho U_b^2} = \frac{\Delta p D_b}{2 \rho U_b^2 L}
\]  

(22)

The present friction coefficients are normalized using the friction coefficient for a smooth passage, given by the Blasius equation:

\[
c_{f0} = 0.079 \Re^{-0.25}
\]  

(23)
The findings to emerge from Figure 5 (a) are similar to those of Figure 4. Both models show $P/k = 9$ to have the highest level of heat transfer. This is not only consistent with the data of Rau et al., but it also confirms the findings of Okamoto et al. [9] who found that $P/k = 9$ was the optimum ratio to augment turbulence intensity and maximize heat transfer levels. However, for all $P/k$ ratios, the $k$-$\varepsilon$ model overpredicts the average heat transfer rate. This overprediction is most severe at $P/k = 9$. The predictions of the $\nu^2f$ model, on the other hand, are in very good agreement with the data.

The normalized streamwise velocity distribution at the plane $y/k = 0.1$ is shown in Figure 6 (a). From this figure it can be seen that the length of the experimentally-determined separation zone downstream of a rib is about $3.5k$ for $P/k = 9$ and 12 and the velocity recovery immediately after the reattachment point, is strongest for $P/k = 9$. The $k$-$\varepsilon$ model indicates the length of the separation zone to be between $2k$ and $2.5k$ for all three $P/k$ ratios, while the $\nu^2f$ model shows this length to be $4.5k$. It is worth mentioning that, although the $\nu^2f$ model underestimates the magnitude of the velocity close to the downstream rib, it correctly indicates that there is no reattachment point for $P/k = 6$.

Also, from Figure 6 (a), one can see that a greater distance between the reattachment point and the downstream rib leads to higher maximum positive velocities at larger $P/k$ ratios. This trend is correctly resolved by both models.

Figure 6 (b) shows the normalized wall-normal velocity, $V/U_b$, at the height of the rib ($y/k = 1$). Simulations for $P/k = 6$, 9 and 12 are shown. Except for $P/k = 12$, the results of the $\nu^2f$ model are in close agreement with the data. However, for $P/k = 12$, the $\nu^2f$ model, in common with the $k$-$\varepsilon$ model, underpredicts the magnitude of the downward vertical motion, especially close to the downstream rib.

While in a smooth channel the measured pressure drop can be directly linked to the shear stress at the wall, this is not true for a ribbed channel. The...
impingement in front of the rib leads to a local high static pressure in this zone, while there is a low pressure zone behind the rib. These effects can be seen in Figure 7, where the pressure coefficient distribution between the two ribs is plotted against $x/k$ for $P/k = 9$. The pressure coefficient is defined as:

$$C_p = \frac{p_s - p_{ref}}{0.5 \rho U_b^2}$$ (24)

As can be seen from Figure 7, except for the recirculation zone, where the predictions of both models are in good agreement with the data, they generally tend to produce too high pressure levels.

4. CONCLUSIONS

Numerical simulations of the flow and heat transfer in 2-dimensional rib-roughened ducts were performed using the Lien-Chen-Leschziner $k$-$\varepsilon$ and Durbin $v^2f$ models. All simulations were undertaken using a commercial CFD package, ‘STAR-CD’. Three configurations with $P/k$ ratios of 6, 9 and 12 were considered. Comparison between the computations and experimental data confirmed that the $k$-$\varepsilon$ model severely overpredicts heat transfer levels, while heat transfer levels generated by the $v^2f$ model were closer to the experimental values. It was also found that $P/k$ = 9 was the optimum ratio to maximize heat transfer performance, while, at the same time, the pressure loss attained its maximum for this $P/k$ ratio. This finding is consistent with the findings of [9] and [10]. In conclusion, a general shortcoming in the quantitative accuracy of the RANS models examined in calculating flow in ribbed channels was identified.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$</td>
<td>Cross-sectional area of the channel</td>
</tr>
<tr>
<td>$c_f$</td>
<td>Local friction coefficient</td>
</tr>
<tr>
<td>$c_p$</td>
<td>Specific heat coefficient at constant pressure</td>
</tr>
<tr>
<td>$C_p$</td>
<td>Pressure coefficient, $(p_s - p_{ref})/(0.5 \rho U_b^2)$</td>
</tr>
<tr>
<td>$D_h$</td>
<td>Hydraulic diameter, $4A/P$</td>
</tr>
<tr>
<td>$k$</td>
<td>Height of the rib or turbulent kinetic energy, $\overline{u'\nu'}/2$</td>
</tr>
<tr>
<td>$L$</td>
<td>Length scale</td>
</tr>
<tr>
<td>$Nu$</td>
<td>Nusselt number, $qD_b/(\lambda(T_w - T_b))$</td>
</tr>
<tr>
<td>$p$</td>
<td>Pressure</td>
</tr>
<tr>
<td>$P$</td>
<td>Pitch; wetted perimeter</td>
</tr>
<tr>
<td>$P_k$</td>
<td>Rate of shear production of $k$, $-\overline{u'\nu'(\partial U_j/\partial x_i)}$</td>
</tr>
<tr>
<td>$p_s$</td>
<td>Static pressure at the wall</td>
</tr>
<tr>
<td>$Pr$</td>
<td>Prandtl number, $c_p \mu / \lambda$</td>
</tr>
<tr>
<td>$\dot{q}$</td>
<td>Wall heat flux</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number, $U_D \delta / \nu$</td>
</tr>
<tr>
<td>$Re_r$</td>
<td>Turbulent Reynolds number, $k^2/(\nu \varepsilon)$</td>
</tr>
<tr>
<td>$T$</td>
<td>Temperature</td>
</tr>
<tr>
<td>$T_\tau$</td>
<td>Turbulent timescale</td>
</tr>
<tr>
<td>$U_\tau, u_\tau$</td>
<td>Mean, fluctuating velocity components in Cartesian tensors</td>
</tr>
<tr>
<td>$x, y$</td>
<td>Streamwise and wall-normal coordinates</td>
</tr>
<tr>
<td>$y^*$</td>
<td>Dimensionless distance from the wall, $yU_\tau/\nu$</td>
</tr>
</tbody>
</table>

Greek Symbols

<table>
<thead>
<tr>
<th>Letter</th>
<th>Symbol</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\varepsilon$</td>
<td>Dissipation rate of $k$</td>
</tr>
<tr>
<td>$\lambda$</td>
<td>Thermal conductivity</td>
</tr>
<tr>
<td>$\mu$</td>
<td>Dynamic viscosity</td>
</tr>
<tr>
<td>$\mu_t$</td>
<td>Turbulent viscosity</td>
</tr>
<tr>
<td>$\nu$</td>
<td>Kinematic viscosity, $\mu / \rho$</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Density</td>
</tr>
<tr>
<td>$\sigma_s$</td>
<td>Turbulent Prandtl number for diffusion of $k$</td>
</tr>
<tr>
<td>$\sigma_\varepsilon$</td>
<td>Turbulent Prandtl number for diffusion of $\varepsilon$</td>
</tr>
<tr>
<td>$\tau_w$</td>
<td>Wall shear stress</td>
</tr>
</tbody>
</table>

Subscripts

<table>
<thead>
<tr>
<th>Subscript</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$b$</td>
<td>Bulk</td>
</tr>
<tr>
<td>$ref$</td>
<td>Reference</td>
</tr>
<tr>
<td>$s$</td>
<td>Static</td>
</tr>
<tr>
<td>$t$</td>
<td>Turbulent</td>
</tr>
<tr>
<td>$w$</td>
<td>Wall</td>
</tr>
</tbody>
</table>

Additional symbols are defined in the text.

ACKNOWLEDGEMENTS

The authors are grateful to the UK Engineering and Physical Sciences Research Council for funding under grant EP/C549465/1, ‘Keeping the Nuclear Option Open’ (KNOO), and wish to acknowledge helpful discussions with J. Gotts and M. Rabbitt of British Energy and also their colleague D. Laurence at the University of Manchester.

REFERENCES


