Developing Turbulent Forced Convection in Two-Dimensional Duct

Y. T. Chen
J. H. Nie
B. F. Armaly
H. T. Hsieh
R. F. Boehm

Department of Mechanical Engineering, University of Nevada, Las Vegas, NV 89154

University of Nevada, Rolla, MO 65401

Department of Mechanical Engineering, University of Missouri-Rolla, Rolla, MO 65401

Department of Mechanical Engineering, University of Nevada, Las Vegas, NV 89154

Developing turbulent forced convection flow in a two-dimensional duct is simulated for Reynolds numbers ranging from 4560 to 12,000. Simultaneously developing velocity and temperature distributions are reported by treating the inlet flow as isothermal with uniform velocity profile. The walls are supplied with uniform heat flux. Distributions of the streamwise and the transverse velocity components exhibit a maximum near the walls, but not at the center of the duct, in the developing region of the flow. The friction coefficient and the Nusselt number do not reach the fully developed values monotonously, and a minimum in their distribution appears in the developing region. Some results are compared with the available data, and very favorable comparisons are obtained. [DOI: 10.1115/1.2740659]

Keywords: turbulent, heat transfer, forced convection, developing flow, duct

Introduction

The fundamental turbulent flow and heat transfer mechanism is of great importance from both scientific and engineering viewpoints because it occurs frequently and plays a predominant role in convective momentum, heat, and mass transfer in many industrial applications, such as compact heat exchangers, gas turbine cooling systems, nuclear reactors, and numerous others. Turbulent forced channel flow, due to its geometric simplicity and fundamental nature to understand the transport mechanism, has been studied extensively from both experimental [1–3] and numerical [4–6] approaches. Wei and Willmarth [2] presented the velocity components for the Reynolds numbers of 3000–40,000 using laser Doppler anemometer. The measurements using hot-film anemometry were reported by Johansson and Alfredsson [3]. During the recent years, the performance of turbulence and heat transfer models in predicting the velocity and temperature fields of relevant industrial flows has become increasingly important. Recent advances in large-scale computers have made it possible to conduct the fundamental studies of turbulent flow numerically at moderate Reynolds numbers. Kim et al. [4] numerically solved the unsteady Navier-Stokes equations using direct numerical simulation (DNS) at Reynolds number of 3300. A number of statistical correlations which were complementary to the existing experimental data were reported. Moser et al. [5] simulated the fully developed channel flow for the friction Reynolds number of 180–590 using direct numerical simulation. Abe et al. [6] reported the effects of Reynolds number on various turbulent statistics for the friction Reynolds numbers of 180, 395, and 640.

These experimental/numerical results are restricted to the fluid mechanics case. Kim and Moin [7] studied the heat transfer (mass transport) in two-dimensional forced convection channel flow for different Prandtl numbers. Kasagi et al. [8] revisited the problems by employing a constant time-averaged heat flux boundary condition on the walls for a mild Reynolds number of 4580. Recently, Debusschere and Rutland [9] investigated the transport of passive heat transfer in a plane channel and Couette flow for the Reynolds number of 3000.

However, most of the published results are limited to the fully developed turbulent channel flow and/or the Reynolds number is mild. To the best of the authors’ knowledge, study of simultaneously developing turbulent convection flow in channel has not been reported in the published literature. This fact, together with the realization that in the practical applications the flow and heat transfer start with the developing region and the Reynolds number is high, motivated the present study.

Problem Statement and Simulation Procedures

Turbulent developing forced convection flow in a parallel channel is numerically simulated. The simulated geometry has a height (h) of 0.01 m. By exploiting the symmetry of the flow and temperature fields in the transverse direction, the computational domain was reduced to half of the actual height of the channel (δ = 0.005 m). This assumption of symmetry is confirmed by computation with a whole domain. The length of the computational domain is 2 m, i.e., x/δ = 400. This computational domain was used in order to obtain the asymptotic behavior for the fully developed regime. The origin of the coordinates system is located at the bottom edge and the inlet of the channel. The steady turbulent Navier–Stokes and energy equations are solved numerically together with the continuity equation using the finite volume method and four-equation low-Reynolds-number model [10,11]. The governing equations based on the approximation of eddy-viscosity for fluid and eddy-diffusivity for heat and the constants appearing in the governing equations can be found in Ref. [12]. They are not shown here due to space limitation. The physical properties are treated as constants and evaluated for air at the inlet temperature (T0) of 20°C, that is, density (ρ) is 1.205 kg/m³, molecular dynamic viscosity (μ) is 1.82×10⁻⁵ kg/(m·s), specific heat (Cp) is 1005 J/(kg·°C), and thermal conductivity (λ) is 0.0258 W/(m·°C). Flow at the inlet section of the channel (x/δ = 0) is considered to be isothermal (T0=20°C), with a uniform streamwise velocity component (u) equal to the bulk velocity (u0).

The other velocity component, v, is set to be equal to zero at that inlet section. No slip boundary condition (zero velocities) is applied to the wall surface. Uniform heat flux (qw=500 W/m²) is applied at the walls. The kinetic energies (k and l) at the wall are equal to zero. The dissipation rate for the fluid field (εk) is set to be εk=2ρσk/ρk and the dissipation rate for the thermal field (εl) is εl=α(∂T/∂n)² at the solid walls, where k is the kinetic energy and n is the normal distance of the first node near the wall, respectively, and α is thermal diffusivity. Fully developed flow and thermal boundary conditions are imposed at the exit section of the computational domain (x/δ=400).
The governing equations are discretized using the staggered grid arrangement, and the resulting finite volume equations using SIMPLE algorithm are solved numerically by making use of a line-by-line method combined with ADI scheme [13]. Nonuniform grid system is employed in the simulations, and the grid is highly concentrated near the solid wall, in order to ensure the accuracy of the numerical simulations. Comparisons of the dimensionless mean velocity \( u^+ = u/u_w \) where \( u_w = \sqrt{T_s/\mu} \) and \( T_s \) is wall shear stress and temperature \( T^+ = (T-T_b)/T_w \), where \( T_w \) is wall temperature and \( T = q_w h/(pC_p \mu) \) profiles with the direct simulated data [8] are shown in Fig. 1. These profiles are taken at the streamwise location of \( x/\delta = 390 \), where the flow and heat transfer have already become fully developed. The mean velocity and temperature profiles agree closely well with those DNS results. Results from several grid densities for a Reynolds number \( \text{Re}_m = 2(\mu u_w'^3/\delta \mu) \) of 6000 were used in developing a grid independence solution for this study. The velocity and temperature values at a selected point in the flow domain are presented in Table 1 for different computational grids. A grid of \( 100(x) \times 41(y) \) is used during the present simulations and a denser grid of \( 120(x) \times 51(y) \) results in less than 1\% difference in the predicted streamwise velocity component at the selected point. The convergence criterion required that the maximum relative mass residual based on the inlet mass be smaller than \( 3 \times 10^{-6} \). It usually takes about 45,000 iterations to meet this requirement. The other test case to validate the flow simulation code that is used in this study is the two-dimensional turbulent flow and heat transfer over backward-facing step [12]. Predicted velocity profiles at several streamwise locations are compared with the laser Doppler measurements [14] as shown in Fig. 2 with good agreement between the predicted and measured data (\( S \) is step height). Comparison of the computed Stanton number \( St = q_w h/(pC_p \mu(T_w-T_b)) \), where \( u_w \) is centerline velocity) on the heated bottom wall is made with the available measured data [14] as shown in Fig. 2. Very close agreement is obtained for the Stanton number profile inside the recirculation region and near the reattachment region, which justifies the present convection flow simulation code and provides with confidence for the next simulations.

**Results and Discussions**

The simultaneously developing turbulent flow and heat transfer in a parallel channel is simulated for different Reynolds numbers of 4560, 6000, 7500, 9000, 10,500 and 12,000. The bulk Nusselt number \( Nu_w = q_w h/\lambda(T_w-T_b) \), where \( T_b \) is bulk temperature] for the fully developed turbulent channel flow with a constant heat flux is shown in Fig. 3. The direct simulated result [8] and the experimental data [15] are also included in Fig. 3. The Nusselt number increases with the increase of Reynolds number. The Nusselt number obtained by the present simulation for \( \text{Re}_m = 4560 \) is 15.4, and it is in excellent agreement with the DNS data which is 15.4 [8].

Distributions of the mean streamwise velocity component (\( u \)) at different streamwise locations in the entrance region of the channel are shown in Fig. 4 for the Reynolds numbers of 4560 and 12,000. Similar results are also obtained for other Reynolds numbers but not presented in the manuscript due to space limitation. Velocity at the centerline of the channel increases from the uniform inlet velocity profile to the fully developed velocity profile. The nonparabolic pattern observed for the laminar flow in a cir-

---

**Table 1**

<table>
<thead>
<tr>
<th>Grid</th>
<th>Size ((x \times y))</th>
<th>( u ) ((m/s))</th>
<th>( v ) ((m/s))</th>
<th>( T ) (^{°C})</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>(40 \times 11)</td>
<td>9.294</td>
<td>0.001381</td>
<td>22.26</td>
</tr>
<tr>
<td>2</td>
<td>(60 \times 21)</td>
<td>9.429</td>
<td>0.002070</td>
<td>22.14</td>
</tr>
<tr>
<td>3</td>
<td>(80 \times 31)</td>
<td>9.457</td>
<td>0.002196</td>
<td>22.10</td>
</tr>
<tr>
<td>4</td>
<td>(100 \times 41)</td>
<td>9.465</td>
<td>0.002218</td>
<td>22.08</td>
</tr>
<tr>
<td>5</td>
<td>(120 \times 51)</td>
<td>9.471</td>
<td>0.002231</td>
<td>22.07</td>
</tr>
</tbody>
</table>

---

**Fig. 1** Comparisons with the DNS data for the mean profiles

**Fig. 2** Comparisons of the mean velocity profiles and the Stanton number for the separated convection flow
cular duct [16] is also seen for the turbulent flow in the parallel channel. In the entrance region, the maximum streamwise velocity component does not appear at the center of the channel, but near the solid wall ($y/\delta=0$). For the Reynolds numbers of 4560 and 12,000, at $x/\delta=1$, their peak values develop at $y/\delta=0.16$ and 0.11, respectively. The similar feature can also be seen at the section of $x/\delta=0.5$. Distributions of the streamwise velocity component in the entrance region are displayed in Fig. 5. The dashed line denotes the locations where the streamwise velocity component ($u$) is a maximum at the streamwise planes. Figure 5 also shows that in the entrance region the maximum of this velocity component does not appear at the center of the channel, but near the wall.

Distributions of the transverse velocity component ($v$) at several streamwise locations for the Reynolds numbers of 4560 and 12,000 are shown in Fig. 6. The dashed line denotes the locations where the transverse velocity component ($v$) is a maximum at the streamwise planes. This velocity component is zero at the wall ($y/\delta=0$) and the symmetry centerline ($y/\delta=1$). For $Re_m=4560$ and 12,000 at $x/\delta=1$, their maximum values develop at $y/\delta=0.12$ and 0.09, respectively. Its magnitude decreases and its location moves toward the centerline as the $x$-planes increase in the streamwise direction. At the same $x$-planes, the magnitudes of the peak $v$-velocity component decrease as the Reynolds number increases. Distributions of the transverse velocity component in the entrance region are presented in Fig. 7. The locations where the transverse velocity component is a maximum in the transverse direction for the $x$-planes are shown in Fig. 7 with the dashed lines. Near the inlet section, the maximum $v$-velocity component develops to appear close to the wall ($y/\delta=0$). Its location moves toward the centerline of the channel as the location of the $x$-plane increases in the flow direction.

Distributions of the temperature difference based on the wall temperature ($\Delta T=T_w-T$) at several streamwise planes are shown in Fig. 8 for the Reynolds numbers of 4560 and 12,000. At the same streamwise planes, the temperature difference decreases as the Reynolds number increases. One feature in the temperature distribution is that the temperature difference increases at first from the inlet section ($x/\delta=0$) to the streamwise section of $x/\delta=20$, then it decreases at the downstream $x$-planes (from $x/\delta=40$ to 100). The results show that profiles of the temperature difference at other downstream $x$-planes ($x/\delta>100$) are almost...
The friction coefficient $C_f = \frac{2\tau_w}{\rho u_0^2}$ for different Reynolds numbers is presented in Fig. 9. The friction coefficient decreases with the increase of Reynolds number. The friction coefficient becomes smaller from the inlet section and reaches the fully-developed values $(x/\delta = 100)$. One feature in its distributions is that decrease of the friction coefficient in the streamwise direction does not follow the monotonous way in the entrance region. A minimum value is observed near the inlet section, which can be seen more clearly for the Reynolds number of 4560. Distributions of the bulk Nusselt number for different Reynolds numbers are shown in Fig. 10. The magnitude of Nusselt number increases with the increase of Reynolds number. The Nusselt number becomes smaller in its magnitude from the inlet section and approaches the fully developed values $(x/\delta > 120)$. Similarly, the Nusselt number in the streamwise direction does not decrease monotonously in the entrance region. A minimum value in the

Fig. 6 Distributions of the transverse velocity component at several $x$-planes

Fig. 7 Distributions of the transverse velocity component $(v)$

Fig. 8 Distributions of the temperature difference $(T_w - T)$ at several $x$-planes

overlaid with the profile at $(x/\delta = 100)$. The similar features are also observed for other studied Reynolds numbers.

The friction coefficient $(C_f = 2\tau_w/\rho u_0^2)$ for different Reynolds numbers is presented in Fig. 9. The friction coefficient decreases with the increase of Reynolds number. The friction coefficient becomes smaller from the inlet section and reaches the fully-developed values $(x/\delta > 100)$. One feature in its distributions is that decrease of the friction coefficient in the streamwise direction does not follow the monotonous way in the entrance region. A minimum value is observed near the inlet section, which can be seen more clearly for the Reynolds number of 4560. Distributions of the bulk Nusselt number for different Reynolds numbers are shown in Fig. 10. The magnitude of Nusselt number increases with the increase of Reynolds number. The Nusselt number becomes smaller in its magnitude from the inlet section and approaches the fully developed values $(x/\delta > 120)$. Similarly, the Nusselt number in the streamwise direction does not decrease monotonously in the entrance region. A minimum value in the
streamwise distribution of the Nusselt number develops in the entrance region of turbulent convention channel flow for the studied Reynolds numbers

Conclusions

Convection in turbulent forced flow in a parallel channel, where the walls are heated with constant heat flux, is examined for the purpose of determining the flow and thermal behavior that simultaneously develops in the entrance region of this geometry. Nonparabolic patterns in the mean streamwise velocity component are observed for the studied Reynolds numbers. The peak streamwise velocity component develops to appear near the wall, but not at the centerline, in the entrance region. Such patterns are also seen in the distribution of temperature field. The friction coefficient decreases with the increase of Reynolds number. The Nusselt number increases with the increase of Reynolds number. It does not reach the fully developed value monotonously and a minimum develops in the developing region.

Acknowledgment

This work was in part supported by U.S. Army Research Laboratory under Grant No. DAAD 19-03-2-0007 and by U.S. Department of Energy under Grant Nos. DE-FG02-03ER46067 and RF-05-HFS-006. J.H.N. gratefully acknowledges H. Lan for useful discussions.

References