Lattice Boltzmann Method for Turbulent Heat Transfer in Wavy Channel Flows

Abstract—The hydrodynamic and thermal lattice Boltzmann methods are applied to investigate the turbulent convective heat transfer in the wavy channel flows. In this study, the turbulent phenomena are modeling by large-eddy simulations with the Smagorinsky model. As a benchmark, the laminar and turbulent backward-facing step flows are simulated first. The results give good agreement with other numerical and experimental data. For wavy channel flows, the distribution of Nusselt number and the skin-friction coefficients are calculated to evaluate the heat transfer effect and the drag force. It indicates that the vortices at the trough would affect the magnitude of drag and weaken the heat convection effects on the wavy surface. In turbulent cases, if the amplitude of the wavy boundary is large enough, the secondary vortices would be generated at troughs and contribute to the heat convection. Finally, the effects of different Re on the turbulent transport phenomena are discussed.

Keywords—Heat transfer, lattice Boltzmann method, turbulence, wavy channel.

I. INTRODUCTION

The lattice Boltzmann equation (LBE) is developed from the lattice gas cellular automata (LGCA) by introducing ensemble-average density distribution function to substitute the Boolean operators in LGCA [1]. In the lattice Boltzmann method (LBM), the fluid flow is treated on a statistical level and calculated by a simplified kinetic model, in which macroscopic physical properties are associated with microscopic processes. This bottom-up scheme is different from traditional top-down schemes of computational fluid dynamics (CFD), which analyze flow fields by solving macroscopic variables such as density, momentum, and energy equation for heat transfer.

Similar to LGCA, LBM performs a two-step evolution of particle distribution functions on a specific lattice model, namely particle distribution “collisions” on lattice nodes and stream “propagations” from one node to all neighbors along the lattice directions. After streaming, new local properties on lattice nodes are obtained by distribution components from neighbors, so that macroscopic properties of fluid flows can be calculated by solving this velocity discrete Boltzmann equation.

In LBE, the complex collision terms of the Boltzmann equation can be replaced by a simplified relaxation model, namely the well-known Bhatnagar-Gross-Krook (BGK) model [7]. By the Chapman-Enskog expansion, this lattice BGK model can be recovered to the Navier-Stokes equations for hydrodynamics or to energy equation for heat transfer.

Simulating turbulent flows by the LBM is an attractive topic [8], [9]. The common methods for simulating turbulence are direct numerical simulation (DNS), Reynolds average numerical simulation (RANS), and large-eddy simulation (LES) [10]. The main idea of the LES is to decompose the turbulence into two parts of different scales, namely a large scale part solved by the Navier-Stokes equations and a small scale part described by a sub-grid scale (SGS) model. The LES has been used to solve many turbulent problems, e.g. simple hydraulic channels [10], gas turbine combustor flow [11], buoyant jets [12], free surface [13], and heat transfer for turbulent flows [14]. The SGS model used in this study is based on the well-known Smagorinsky model including vortex-viscous and vortex-diffusive forms [15]. Hou et al. [16] used LBM coupled with the standard Smagorinsky model to simulate two-dimensional driven cavity flow at Re up to 100,000. By solving the vorticity-stream function equations with the LES and Smagorinsky model, Chen [17] presented a lattice Boltzmann algorithm to simulate turbulent driven cavity flow. The numerical stability of this model is better than the traditional LBM, which solves mass density, pressure, and velocity for Navier-Stokes equations and has constraints of low Mach number and relaxation time. Guan et al. [18] introduced the dynamics SGS model and the dynamical system SGS model for the LBM to solve three dimensional high Re turbulent driven cavity flows. Results were compared with those obtained by the Smagorinsky model.

This paper adopts the hydrodynamic and thermal LBM algorithms to study the transport phenomena of the wavy channel, which is often employed to enhance the heat transfer efficiency in industrial processes [19]. Wang and Chen [20] used a simple coordinate transformation for the complex wavy boundary and solve the stream function, vorticity and energy equations of the fluid flow by the spline alternating-direction implicit method. The effects of the wavy geometry, Reynolds number (Re), and Prandtl number (Pr) on the skin-friction and...
Nusselt number (Nu) were discussed. By the finite element method, Alawadhi [21] studied the steady fluid flow and heat transfer in a wavy channel of 25< Re<1000 with a linearly increasing waviness at the entrance region.

Since turbulent flows are common in nature and engineering, the present study focuses on the turbulent transport phenomena of fluid flow in wavy channels. The LES method is adopted for the LBM simulation of turbulence. As a benchmark, the laminar and turbulent backward-facing step flows are simulated first to validate the feasibility of this LBM-LES method for an open flow system. The result of the reattachment length is compared with other experimental and simulation data [22]–[25]. For wavy channels, the effects of wave geometry, Reynolds number and Prandtl number on hydrodynamic and heat transfer phenomena are discussed simultaneously. The magnitude of drag and heat transfer efficiency are evaluated by inspecting the skin-friction coefficient (Re C_l) and the Nusselt number on the wavy surface.

II. NUMERICAL METHODS

A. Lattice Boltzmann Hydrodynamic Model

In the LBM simulation of incompressible flows, the continuity equation and Navier-Stokes equations are solved by calculating the evolution of density distribution function according to the LBE on the lattice model. The Boltzmann equation with the BGK model can be written as follows:

\[
\frac{\partial f}{\partial t} + \mathbf{v} \cdot \nabla f = \frac{1}{\tau_f} \left( f^{eq} - f \right)
\]  

(1)

where \(\mathbf{v}\) is microscopic velocity, and \(\tau_f\) is the relaxation time for the density distribution function, \(f\), towards the local equilibrium. The equilibrium distribution function \(f^{eq}\) is related to the Maxwell-Boltzmann equilibrium distribution. By applying a lattice model of a discrete velocity set, (1) is transformed into the discrete LBGK form and discretized in the time and space domain as

\[
f \left( \bar{x} + \bar{e}_a \Delta t, t + \Delta t \right) - f \left( \bar{x}, t \right) = -\frac{1}{\tau_f} \left[ f \left( \bar{x}, t \right) - f^{eq} \left( \bar{x}, t \right) \right]
\]  

(2)

where \(f \left( \bar{x}, t \right)\) and \(\bar{e}_a\) are the component of density distribution function and lattice velocity vector in \(\alpha\) direction of the lattice model. Equation (2) represents that in each time step, the stream and collision steps are performed individually at nodes with different lattice directions Therefore, this algorithm is suitable for parallel computation.

In the LBM simulation, the common lattice model is the DnQb model [26], which refers to the number \(n\) of dimensional sublattices and to the discrete number \(b\) of spatial translation lattice vectors. This present work adopts the D2Q9 lattice model [6] for both the hydrodynamic and thermal analysis of flows. The discrete velocity set of D2Q9 model in Fig. 1 is defined as

\[
\begin{align*}
\bar{e}_0 &= (0, 0) \\
\bar{e}_{1,3,5,7,8} &= (\pm c, 0), (0, \pm c) \\
\bar{e}_{2,4} &= (\pm c, \pm c)
\end{align*}
\]  

(3)

where \(c = \Delta x/\Delta t = \Delta y/\Delta t\) is the lattice streaming speed related to the run time step, \(\Delta t\), and the grid spacing \(\Delta x\) and \(\Delta y\). The density equilibrium distribution function is given by

\[
f^{eq} = \rho \omega_a \left[ 1 + \frac{\bar{e}_a \cdot \bar{u}}{c_s^2} + \frac{(\bar{e}_a \cdot \bar{u})^2}{2c_s^4} - \frac{u^2}{2c_s^2} \right]
\]  

(4)

where \(\bar{u} \left( \bar{x}, t \right)\) is the velocity vector at the lattice node of position \(\bar{x}\). The weight parameters are \(\omega_0 = 4/9\), \(\omega_{2,3,4} = 1/9\), and \(\omega_{5,6,7,8} = 1/36\) respectively. The local density and velocity are obtained by

\[
\rho = \sum_a f_a
\]

(5)

\[
\bar{u} = \frac{1}{\rho} \sum_a f_a \bar{e}_a
\]

(6)

Fig. 1 D2Q9 model for LBM simulation

By the Chapman-Enskog expansion, this LBGK model can be recovered to the governing equations of incompressible flow at low Mach number with the pressure, \(p = \rho c_s^2\), related to the lattice sound speed, \(c_s = c/\sqrt{3}\). The kinematic viscosity is given by \(\nu = c_s^2 \left( \tau_f - 0.5 \right)\Delta t\).

B. Lattice Boltzmann Thermal Model

Similar to the hydrodynamic model, the macroscopic energy equation can be derived from a LBM algorithm for the temperature distribution function by Chapman-Enskog expansion. The thermal LBM model used in this paper is proposed by Shan [27], which is based on the model for simulating flows with multi-phases and -components [28]. In this model, the heat transfer of fluid flow is calculated by the evolution equation of temperature distribution function, \(g_a \left( \bar{x}, t \right)\), as
\[ g_a (\tilde{x} + \tilde{e}_a \Delta t, t + \Delta t) - g_a (\tilde{x}, t) = - \frac{1}{\tau_g} \left[ g_a (\tilde{x}, t) - g_a^e (\tilde{x}, t) \right] \]  

where \( \tau_g \) is the relaxation time. The temperature equilibrium distribution function for the D2Q9 lattice model is given by

\[ g_a^e = T \omega_a \left[ 1 + \frac{\bar{u}_a - \bar{u}}{c_s^2} + \left( \frac{\bar{e}_a - \bar{u}}{2c_s^2} \right)^2 - \frac{n^2}{2c_s^2} \right] \]  

The local macroscopic temperature is calculated as

\[ T = \sum_a g_a \]  

The diffusivity is \( \kappa = c_s^2 (\tau_g - 0.5) \Delta t \).

**C. LBM-LES Method**

In the large-eddy simulation, the physical quantity \( \Phi \) is decomposed into large and small scale parts by the filtering process as follows [10]

\[ \tilde{\Phi} = \frac{1}{\Delta} \int_{\Delta} \frac{1}{\Delta} \Phi (\tilde{x}, t) d\tilde{x} \]  

where \( \Phi \) is the large scale physical quantity after filter process, and \( \Delta \) is the filter length equal to the mesh length. The spatial filter function, \( G \), is the box filter function as follows,

\[ G (\tilde{x} - \tilde{x}) = \begin{cases} 1 & \delta (\tilde{x} - \tilde{x}) \leq 0.5 \Delta \\ 0 & \delta (\tilde{x} - \tilde{x}) > 0.5 \Delta \end{cases} \]  

By filtering the density and temperature distribution function, the hydrodynamic and thermal LBM-LES models with the same forms as before filtering can be obtained [16].

In the LBM-LES model, the viscosity are replaced by the equivalent quantity, \( \nu_{\text{total}} = \nu + \nu_t \). By applying the Smagorinsky model, the eddy viscosity, \( \nu_t \), is represented as

\[ \nu_t = (C_s \Delta)^2 \tilde{S} \]  

where \( C_s \) is the Smagorinsky constant of a positive value. In this paper, the Smagorinsky constant is fixed as \( C_s = 0.1 \). The magnitude of strain rate tensor \( \tilde{S} \) is calculated by

\[ \tilde{S} = \sqrt{\frac{\tau_f^2 + 18 \sqrt{2} (C_s \Delta)^2 Q^2}{\rho} - \tau_f} \]  

where \( Q \) represents the magnitude of the non-equilibrium stress tensor, \( \Pi \), and is calculated as

\[ Q = \Pi \Pi_\rho \]  

\[ \Pi_\rho = \sum_n e_n e_n (f_n - f_n^e) \]

Similarly, the equivalent diffusivity, \( \kappa_{\text{total}} = \kappa + \kappa_t \), is used for the thermal LBM-LES model with introducing the eddy diffusivity, \( \kappa_t \). For a fluid flow of specific Reynolds number, \( Re = UL / \nu \), and Prandtl Number, \( Pr = \nu / \kappa \), and \( \kappa \) can be calculated according to the characteristic velocity and length of the flow field. The eddy diffusivity is also obtained by defining the sub-grid Prandtl Number as \( Pr_t = \nu_t / \kappa_t \).

**D. Boundary Conditions and Convergence Criteria**

In the LBM simulation, the boundary conditions are implemented by calculating the unknown components of distribution function on boundary nodes after streaming process. These unknowns are due to no propagation from the boundary into the flow and can be solved by applying appropriate method. In this study, the no-slip boundary condition of the stationary wall is realized by the unknown-index algorithm [6] and the common bounce-back rule [29]. The boundary condition of a given velocity or temperature is implemented by the forced-equilibrium method [16] as

\[ f_n (x_j, t + \Delta t) = f_n^e (x_j, t) \]  

\[ g_n (x_j, t + \Delta t) = g_n^e (x_j, t) \]

where \( x_j \) represents the position at the boundary node.

For laminar flows, the criteria of convergence are as follows,

\[ \sum_{i,j} \left| \tilde{V} (x_{i,j}, t + \Delta t) - \tilde{V} (x_{i,j}, t) \right| \leq 1.0 \times 10^{-7} \]

\[ \sum_{i,j} \left| \tilde{T} (x_{i,j}, t + \Delta t) - \tilde{T} (x_{i,j}, t) \right| \leq 1.0 \times 10^{-7} \]

In the turbulence, the unsteady flow phenomena are captured by evaluating the time average of physical quantities.

**III. RESULTS AND DISCUSSION**

In the LBM-LES simulation, all equations and physical quantity are non-dimensionalized by the lattice length, lattice speed, and reference density, \( \rho_0 \). The dimensionless temperature is defined as \( T^* = (T - T_\infty) / (T_w - T_\infty) \). The numerical simulation is confirmed to be grid independent.
A. Benchmark

Fig. 2 is the schematic view of the backward-facing step flow. The expansion ratio and the Reynolds number are defined as
\[ ER = \frac{H}{h} = 2 \] and
\[ Re = \frac{U_{in} H}{\nu} \] with the average velocity of the fully-developed inlet flow, \( U_{in} = 0.1 c \). This inlet flow with a constant temperature at \( T = T_{in} \) is placed at a distance of \( 40h \) from the step. At the outlet of \( L_{out} = 100h \), the flow with a fixed reference pressure is assumed to be fully developed, i.e. \( \partial U/\partial x = \partial V/\partial x = \partial T/\partial x = 0 \). All stationary walls adopt no-slip boundary conditions and a high constant temperature \( T = T_w > T_{in} \). The Prandtl number and subgrid Prandtl Number are fixed to 1.0 and 0.7.

In the backward-facing step flow, the separation and reattachment phenomena produce the recirculation region downstream the step. As \( Re \) increases, the main recirculation region downstream the step is enlarged accompanying with a secondary recirculation region produced near the upper wall. Fig. 3 shows the reattachment length of main recirculation region in laminar flows. The present results are close to other numerical and experiment results [22]–[24]. In turbulent backward-facing step flows, more small transient vortices behind the step are formed. The secondary vortex near upper wall is also smaller than that of laminar flow. Armaly et al. [22] indicated the main reattachment length almost kept constant at \( Re \geq 6,600 \). The results by LBM-LES model is listed in Table I with comparison to Armaly et al. [22] and Jongeblod’s work [25].

B. Transport Phenomena in Wavy Channel Flows

Fig. 6 shows the schematic view of the wavy channel flow in this study. The shape of the upper wavy-wall profile is given as
\[ S(x) = L + a \sin \left[ \pi (x - L_{in}) / L \right] \] (20)

The amplitude-wavelength ratio \( a \) is defined as \( a = a/L \), and the Reynolds number is defined as \( Re = U_{in} L / \nu \). As in the backward-facing step flows, the cold fluid of \( T = T_{in} \) flows through hot wavy surfaces of \( T = T_w > T_{in} \) and turns into a fully developed flow at the outlet with a fixed reference pressure. The flat walls are also given a high temperature at \( T_w \) in turbulent
cases, but considered to be adiabatic in laminar cases for comparison with Wang and Chen’s results [20]. In addition, all flat and wavy boundaries are no-slip.

To estimate the heat transfer effect and the magnitude of drag in the wavy channel flow, the Nusselt number (Nu) and the skin-friction coefficient ($\Re C_f$) on the lower wavy boundary (G-F) are calculated. The local Nusselt number is defined as

$$\text{Nu}_z = h_z H / K_f = -\left(\frac{\partial T}{\partial n}\right) L / (T_w - T_o)$$  \hfill (21)

where $h_z$ and $K_f$ are heat transfer coefficient and thermal conductivity respectively. The temperature gradient with respect to the orientation normal to the surface is calculated by

$$\frac{\partial T}{\partial n} = \sqrt{\left(\frac{\partial T}{\partial x}\right)^2 + \left(\frac{\partial T}{\partial y}\right)^2}$$  \hfill (22)

The skin-friction coefficient is defined as $C_f = \tau_w / (\rho U_o^2)$. The shear stress on the surface given by

$$\tau_w = \mu \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)_{y=S(z)}$$  \hfill (23)

Yields

$$\Re C_f = \frac{H}{U_o} \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)_{y=S(z)}$$  \hfill (24)

Fig. 7 is the distribution of skin-friction coefficients and Nusselt number in the laminar wavy channel flow of Pr= 6.93 at Re=500. The results present the same tendency with the results by Wang and Chen [20] except for the skin-friction coefficients at wave troughs in the cases of $\alpha = 0.2$. As fluid flowing into the wave troughs, the flow phenomena are restrained in the recirculation regions with low velocities. These would weaken the heat convection effects and give a negative skin-friction coefficient. On the contrary, fluids in the sections of wave crests are accelerated because of the narrowed passages in a symmetric wavy channel. Therefore, drag forces upwind the wave slopes are intensified and increase the skin-friction coefficients on the wave crests. These phenomena also contribute to the convective heat transfer effect of the wavy surface and give peaks of the Nusselt number on the crests. As shown in Fig. 7, the maximum skin-friction coefficient and Nusselt number occur on the first wave crest, which is the entrance of the wavy sections in the channel, and then decay.
downstream. The case of $\alpha = 0$ represents the fluid flowing through a smooth channel in which the skin-friction coefficient and Nusselt number are decreased fast. Naturally, larger amplitude causes a greater influence on not only drag forces but also heat convection effects. In the case of $\alpha = 0.2$, a small vortex is formed behind the last wavy surface and also causes negative skin-friction coefficients.

In the simulation of turbulence, the Prandtl number is fixed as Pr=1 with a subgrid Prandtl Number of 0.7. The average streamlines and isotherms of the turbulent wavy channel flow at Re=5000 are shown in Figs. 8 and 9. In the case of $\alpha = 0.1$, the structures of transient vortices at wavy troughs are the same as in laminar cases, and hence the tendency of skin-friction coefficient and Nusselt number are similar to those in the laminar cases except for the magnitude, as shown in Fig. 10. In turbulence, the heat convection effects are enhanced greatly by the fast and perturbed flow phenomena as well as the magnitude of drag force on the wavy surface. Although the Prandtl number in the turbulent cases is smaller than in laminar cases, the convection effects give larger and no-decayed Nusselt number on every wave crest, as shown in Fig. 10 (b).

As the amplitude increases to $\alpha = 0.2$, the vortices in the wave troughs are enlarged accompanying with small secondary ones formed near the surface. Fig. 10 indicates that these secondary vortices of the same velocity direction as flows upstream can cause the negative skin-friction coefficients at troughs changing into positive values inversely. Furthermore, the convection effects at the troughs are also improved. Fig. 11 is the distribution of skin-friction coefficient and Nusselt number for the turbulent case of $\alpha = 0.2$ at different Reynolds number. The results show the magnitude of drag and heat convection in turbulence of the wavy channel flow is reduced as Re decreases. Actually, the effects of the secondary vortices at troughs are weak in the case of Re=3000.

**REFERENCES**

Dr. Hsin-Yi Lai received his Ph.D. from the University of Wisconsin at Madison in 1984. He is currently working in National Cheng Kung University as a full-time faculty member. Dr. Lai’s research interests include engineering modeling of various micro-macro systems, design of experiments, and universe exploration. He had experiences in the application of modeling techniques for many different engineering systems. Dr. Lai is currently active in the professional societies of ASME, SME, ASEE and CSME.

Dr. Shing-Cheng Chang received his Ph.D. in Mechanical Engineering from National Cheng Kung University in Taiwan, 2010. He is a postdoctoral fellow of Department of Mechanical Engineering, National Cheng Kung University in Taiwan. His major research areas include computational fluid dynamics and nanotechnology. He has published 20 research papers in international journals, books, and conferences.