International Journal of Heat and Mass Transfer 66 (2013) 338–348

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ijhmt

Application of the lattice Boltzmann method combined with large-eddy simulations to turbulent convective heat transfer

Shing-Cheng Chang, Yue-Tzu Yang, Cha'o-Kuang Chen *, Wei-Lin Chen

Department of Mechanical Engineering, National Cheng Kung University, Tainan, Taiwan

ARTICLE INFO

Article history:
Received 28 December 2012
Received in revised form 24 June 2013
Accepted 27 June 2013

Keywords:
Lattice Boltzmann method
Large-eddy simulation
Turbulence
Heat transfer

ABSTRACT

In this paper, the large-eddy simulation is introduced into the lattice Boltzmann method to study convective heat transfer in turbulent flows. The simulations include a closed lid-driven cavity flow and a backward-facing step flow in both laminar and turbulent regions. The results show that by combining with large-eddy simulations, the lattice Boltzmann method can simulate turbulent flow phenomena well and give good agreement with other experimental and numerical results, while the traditional lattice Boltzmann method fails. Quaternary vortices of the turbulent cavity flows are captured in the simulations as well as the transient vortices of backward-facing step flows. By calculating the distribution of skin-friction coefficients and Nusselt number on the lower wall, the drag and heat transfer efficiency of backward-facing step flows are found to be influenced by the vortices generated near walls significantly, no matter the flow is laminar or turbulent. For laminar cases, the flow phenomena are also greatly affected by the Reynolds number. But in turbulence, the flow field is fully perturbed and chaotic, so that the transport phenomena are approximately independent of the Reynolds number.

1. Introduction

The lattice Boltzmann method (LBM) [1,2] has been developed over twenty years and achieved considerable success in many problems. On the basis of the microscopic nature, the LBM simulates fluid flows by performing a bottom-up scheme, which treats the fluid particles on a statistical level. Simplified kinetic models incorporating essential physics of microscopic processes are constructed in the LBM. Fluid flows are tracked through the evolution of one-particle phase space distribution functions and associated macroscopic averaged properties. Based on the gas kinetic theorem, LBM simulations include two steps, namely particle distribution "collisions" on lattice nodes and stream "propagations" from one node to all neighbors along the lattice directions. After streaming, new distribution components on lattice nodes are obtained from neighbors in a new time step and cause new local and macroscopic properties. Essentially, this is different from other traditional CFD methods, which analyze flow fields by solving macroscopic variables in the Navier–Stokes equations.

As a promising method of computational fluid dynamics (CFD), many difficult problems in traditional CFD can be solved by LBM, e.g. multiphase fluid flows [3], heat transfer [4], microfluidics [5], fluid flows through porous media [6] and fractal geometry [7]. In nature and engineering, turbulent flows are very common. But they are difficult to be solved by theoretical or numerical schemes because of the complicated and irregular characteristics. Therefore, to simulate turbulent flows by the LBM is an attractive topic [8]. Chen and Doolen [9] summarized the lattice Boltzmann method for fluid flows, which included the development of LBM for simulating turbulence. The typical methods for simulating turbulence are direct numerical simulation (DNS), Reynolds average numerical simulation (RANS), and large-eddy simulation (LES), which was proposed first by Deardoff [10]. The base ideal of LES is to decompose the turbulent flow field into large and small scale parts. The large scale part is solved by Navier–Stokes equations, while the small scale part is solved by sub-grid scale (SGS) model.

The SGS model used in this study is based on the well-known Smagorinsky model [11], which includes vortex-viscous and vortex-diffusive forms. Hou et al. [12] used LBM coupled with the standard Smagorinsky model and introduced the eddy relaxation time to simulate two-dimensional driven cavity flow. The Reynolds number was considered up to 100,000. Chen [13] also used a large-eddy-based LBM to simulate turbulent driven cavity flow. This model is corresponding to vorticity-stream function equations and hence has better numerical stability than the traditional LBM, which solves mass density, pressure and velocity for Navier–Stokes equations. Guan and Wu [14] introduced two subgrid models, namely the dynamics SGS model and the dynamical system SGS model, to the lattice Boltzmann method for solving

* Corresponding author. Tel.: +886 6 2757575x62140.
E-mail address: ckchen@mail.ncku.edu.tw (C.-K. Chen).

0017-9310/$ - see front matter
http://dx.doi.org/10.1016/j.ijheatmasstransfer.2013.06.067

three dimensional high Re turbulent driven cavity flows. Results were compared with those obtained from the Smagorinsky model and direct numerical simulation for the same cases.

Although the large-eddy simulation was proposed and applying to study turbulence for decades, the combination of the lattice Boltzmann method and LES were extensively adopted only in recent years. To our knowledge, there are a few studies concerning solving convection heat transfer in turbulence, and most of these studies were focused on the topic of a closed cavity [15–20]. Wu et al. [21] simulated the turbulent heat transfer in a channel flow. Although their work included three dimensional effects, the channel walls were two straight plates. The objective of this study is to apply the LBM by combining with the LES to simulate turbulence. The simulations include the lid-driven cavity flow and the backward-facing step flow. In addition, the turbulent heat transfer phenomena in the backward-facing step flow are simulated and discussed.

For the backward-facing step flow, the separation and reattachment phenomena produce the recirculation region downstream the step. Armaly et al. [22] used the laser-Doppler measurements to observe velocity distribution and reattachment length of a single backward-facing step in a channel. The numerical method was also applied to simulate this problem. The discussions were presented for laminar, transitional and turbulent flow of air in a Reynolds number range of 70 < Re < 8,000. The results indicated that the flow can be considered as laminar for Re < 1,200 and turbulent for Re > 6,600. For the range of 1,200 < Re < 6,600, the flow phenomena are transitional and present noticeable three dimensional effects. Since the present simulations are considered two dimensional cases, this work focuses on the laminar and turbulent regions of the backward-facing step flow only. The reattachment length for the laminar region is also compared with numerical results of Erturk [23] and Ma et al. [24]. For the turbulent region, Jongebloed's numerical results [25] by FULENT with RANS method for turbulence are adopted for validation. Otherwise, the drag effects and heat transfer efficiency are evaluated by calculating the skin-friction coefficients and the Nusselt number, respectively. Finally, the turbulent convective heat transfer phenomena of the backward-facing step flow are discussed in this paper.

2. Numerical methods

2.1. Hydrodynamic model for LBM

The Boltzmann equation with a linearized collision operator can be written as follows:

$$\frac{\partial f}{\partial t} + \vec{v} \cdot \frac{\partial f}{\partial \vec{x}} = \frac{1}{\tau_f}(f^{eq} - f), \tag{1}$$

where $\vec{v}$ is microscopic velocity. The relaxation term on the right side is the simplified collision operator, namely the Bhatnagar–Gross–Krook (BGK) approximation model [26], with the relaxation time τ_f 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, the Boltzmann equation with BGK model (LBGK) can be transformed into the discrete form of

$$\frac{\partial f_\alpha}{\partial t} + \vec{e}_\alpha \cdot \nabla f_\alpha = -\frac{1}{\tau_f}(f_\alpha - f_\alpha^{eq}), \tag{2}$$

where $f_\alpha(\vec{x}, t)$ and $\vec{e}_\alpha$ are the component of density distribution function and lattice velocity vector in the α direction of the lattice model. For evolution of distribution function, this discrete lattice Boltzmann equation can be discretized in the time and space domain as

$$f_\alpha(\vec{x} + \vec{e}_\alpha \Delta t, t + \Delta t) - f_\alpha(\vec{x}, t) = -\frac{1}{\tau_f}[f_\alpha(\vec{x}, t) - f_\alpha^{eq}(\vec{x}, t)]. \tag{3}$$

The LBGK model can be decomposed as two steps repeated for each time step, namely stream and collision steps. These steps are performed individually in different lattice directions according to the specified lattice model.

In the LBM simulation, the DnQb lattice model proposed by Qian et al. [27] of n dimension and b lattice velocities are often used. This study adopts the convenient D2Q9 lattice model [4,7] for both the hydrodynamic and thermal analysis of flows. The discrete velocity set of D2Q9 model is shown in Fig. 1. The velocity distribution are defined as

$$\vec{e}_\alpha = \begin{cases} (0,0) & ,\alpha = 0 \\ c\left[\cos\left((\alpha-1)\frac{\pi}{2}\right), \sin\left((\alpha-1)\frac{\pi}{2}\right)\right] & ,\alpha = 1\text{–}4 \\ \sqrt{2}c\left[\cos\left((2\alpha-1)\frac{\pi}{4}\right), \sin\left((2\alpha-1)\frac{\pi}{4}\right)\right] & ,\alpha = 5\text{–}8 \end{cases} \tag{4}$$

where $c = \Delta x/\Delta t = \Delta y/\Delta t$ is the lattice streaming speed defined by time step Δt and the grid spacing Δx and Δy. The density equilibrium distribution function is given by

$$f_\alpha^{eq} = \rho\omega_\alpha\left[1 + \frac{\vec{e}_\alpha \cdot \vec{u}}{c_s^2} + \frac{(\vec{e}_\alpha \cdot \vec{u})^2}{2c_s^4} - \frac{u^2}{2c_s^2}\right], \tag{5}$$

where $\omega_\alpha = \frac{4}{9}$ for $\alpha = 0$, $\omega_\alpha = \frac{1}{9}$ for $\alpha = 1$–4, and $\omega_\alpha = \frac{1}{36}$ for $\alpha = 5$–8. $\vec{u}(\vec{x}, t)$ is the velocity vector at the lattice node of position $\vec{x}$. The macroscopic density and velocity are calculated as

$$\rho = \sum_\alpha f_\alpha, \tag{6}$$

$$\vec{u} = \frac{1}{\rho}\sum_\alpha f_\alpha \vec{e}_\alpha. \tag{7}$$

By the Chapman–Enskog expansion [2], the LBGK model can be recovered to the macroscopic equations as follows

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho\vec{u}) = 0, \tag{8}$$

$$\frac{\partial}{\partial t}(\rho\vec{u}) + \nabla \cdot (\rho\vec{u}\vec{u}) = -\nabla p + \nabla \cdot \left\{\rho\nu\left[\nabla\vec{u} + (\nabla\vec{u})^T\right] - \frac{\nu}{c_s^2}\nabla \cdot (\rho\vec{u}\vec{u}\vec{u})\right\}, \tag{9}$$

where $p = \rho c_s^2$ is the pressure related to the lattice sound speed, $c_s = c/\sqrt{3}$. The kinematic viscosity is

$$\nu = c_s^2\left(\tau_f - \frac{1}{2}\right)\Delta t. \tag{10}$$

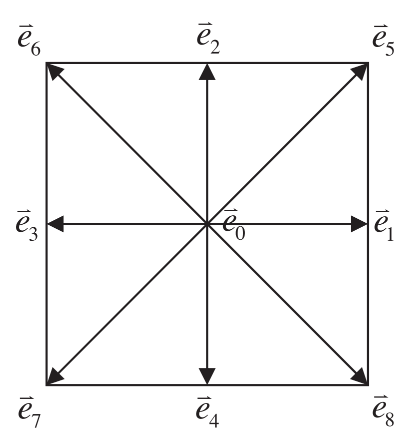

Fig. 1. D2Q9 model for LBM simulation.