International Communications in Heat and Mass Transfer 68 (2015) 188–199

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

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

An efficient smoothed profile-lattice Boltzmann method for the simulation of forced and natural convection flows in complex geometries☆

Yang Hu [a], Decai Li [a,*], Shi Shu [b], Xiaodong Niu [c]

[a] School of Mechanical, Electronic and Control Engineering, Beijing Jiaotong University, Beijing 100044, PR China
[b] School of Mathematics and Computational Science, Xiangtan University, Xiangtan 411105, PR China
[c] College of Engineering, Shantou University, Shantou 515063, PR China

ARTICLE INFO

Available online 21 August 2015

Keywords:
Smoothed profile method
Lattice Boltzmann method
Natural convection
Forced convection

ABSTRACT

In the present study, a numerical method based on the lattice Boltzmann method and smoothed profile method is proposed to simulate the forced and natural convection flows in complex geometries. The solid–fluid boundaries are replaced by a smoothed continuous interface with a finite thickness. A concentration function is used to identify the fluid and solid regions. The flow and temperature fields in different regions are solved on fixed grids. The force term and heat source/sink are added in evolution equations to impose the velocity and temperature boundary conditions. The present approach is validated by some thermal flows examples: flow around a heated circular cylinder, natural convection in a square cavity with a heated circular cylinder, natural convection in a concentric annulus. The computed results show good agreement with the previous data. The high efficiency of the present method is also verified.

1. Introduction

Forced and natural convection flows in complex geometries exist widely in many engineering fields, and it is frequently investigated numerically by many researchers. The treatment of the complex boundaries is a key issue in CFD. Generally speaking, there exist two different mesh techniques: the conforming mesh methods and non-conforming mesh methods. The conforming mesh methods, such as body-fitted method and unstructured method, which require meshes conforming the interface, usually need remesh when dealing with moving boundaries. It may require high computational overhead. Conversely, in the nonconforming mesh methods, the fixed grid is used and the boundary conditions are treated as constraints on the governing equations. These methods avoid the mesh update in numerical procedure and improve the computational efficiency greatly.

In recent years, four different non-conforming mesh methods are very popular: immersed boundary method (IBM) [1,2], immersed interface method (IIM) [3–5], distributed Lagrange multiplier/fictitious domain method (DLM/FDM) [6,7], and smoothed profile method (SPM) [8–10].

The IBM is first introduced by Peskin when he studied the blood flow in the heart. In IBM, two types of meshes are used. A fixed Cartesian mesh is used for the fluid and the boundary is represented by a set of Lagrangian points. The interaction between fluid and solid is computed in terms of distribution and interpolation operations using the smoothed Dirac delta function. After Peskin's pioneering work on IBM, many subsequent studies have been done to improve and extend this method. The IIM is developed by Li and co-workers which incorporates the interface jump conditions into finite difference schemes [3–5]. Compared with the IBM, IIM has higher accuracy. In IIM, the boundary force is used to construct the interface jump conditions in the pressure and the derivatives of the velocity. The DLM/FDM was firstly proposed by Glowinski to study the incompressible particulate flows [6,7]. The main features of DLM/FDM are that the governing equations are discretized in space on an extended domain, and the boundary conditions on the original domain can be enforced by using the Lagrange multiplier approach.

Smoothed profile method (SPM) is another efficient non-conforming mesh method. Nakayama and Yamamoto had done an initial work on SPM [8]. In contrast to the above methods, the solid–fluid boundaries are replaced with a continuous interface by assuming a smoothed profile in SPM. And a concentration function is introduced to describe the flow and solid regions. The effect of this concentration function is similar to the level set function in level set method [11]. Different from the above non-conforming mesh methods, SPM does not need to use the marker points to represent the inner boundary of the solid region and all calculations are implemented on the Eulerian nodes. So it avoids the computational cost of interpolations and distributions operations. As same as the other non-conforming mesh methods, an external force

☆ Communicated by W.J. Minkowycz

* Corresponding author.
E-mail addresses: hy304@126.com (Y. Hu), dcli@bjtu.edu.cn (D. Li).

http://dx.doi.org/10.1016/j.icheatmasstransfer.2015.05.030
0735-1933/

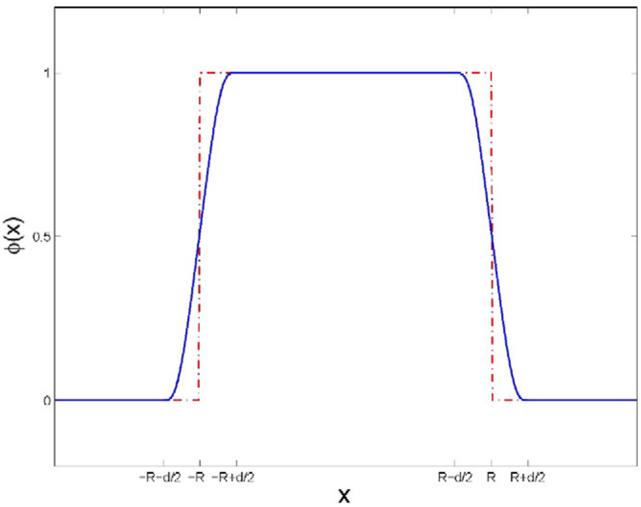

Fig. 1. The schematic of the smoothed profile (Blue line).

term is added to impose the solid–fluid impermeability condition. Nakayama and Yamamoto have applied SPM successfully to simulate colloidal dispersions system and investigate the electrohydrodynamic effects [8,12]. After the initial work of Nakayama and Yamamoto, many related studies have been carried out. Luo et al. proposed a high-order semi-implicit SPM and analyzed its numerical error [9]. Then Luo and Suk extended their method to resolve the electrokinetic flows [13]. Kang et al. proposed a so-called one-stage SPM to simulate the suspended paramagnetic particulate flows [14].

It is noteworthy that the lattice Boltzmann method (LBM) has received many attentions in recent years due to its high efficiency. Different from conventional fluid solver based on Navier–Stokes equations, LBM is derived from kinetic theory [15]. In LBM, an algebraic equation is solved on a uniform Cartesian grid. Via simple streaming and collision steps, the complex fluid systems can be simulated. LBM has been applied in multicomponent and multiphase flows [16], micro-flows [17], turbulent flows [18] and fluid–solid interactions [19–22]. LBM has also been used to investigate the forced and natural convection flows in complex geometries [23–34]. The computational approach coupled LBM and non-conforming mesh methods have been studied by many scholars. Feng and Michaelides proposed an immersed boundary-lattice Boltzmann method (IB-LBM) for solving fluid-particles interaction problems [35]. Shi and Phan-Thien considered a coupling algorithm based on DLM/FDM and LBM to deal with the fluid/elastic–solid interactions [36]. Jafari et al. presented a numerical scheme which couples the LBM and SPM and they successfully simulated particulate suspensions using this method [10].

In this paper, our purpose is to construct a thermal smoothed profile-lattice Boltzmann method (SP-LBM) to simulate the forced and natural convection flows in complex geometries. Although SPM is a very high efficiency method, to best of our knowledge, there are not any works to study the thermal flows using this method. In the present method, the double-distribution-function lattice Boltzmann method is employed. A heat source/sink is introduced to enforce the temperature boundary condition. Clearly, the present thermal SP-LBM still keeps the advantage of the high efficiency. Some forced and natural convection problems are simulated to validate the present method.

2. Numerical schemes

2.1. Fluid and temperature fields solver

In our study, the double-distribution-function lattice Boltzmann method is chosen as fluid and temperature fields solver. It is an efficient alternative technique of the Navier–Stokes solver. The passive-scalar thermal model which is proposed by Shan is used to handle the temperature field [37]. The evolution equations of double-population LBM with source term can be written as

$$f_\alpha(\mathbf{x}+\mathbf{e}_\alpha\Delta t, t+\Delta t)-f_\alpha(\mathbf{x},t) = -\frac{1}{\tau_f}\left(f_\alpha(\mathbf{x},t)-f_\alpha^{eq}(\mathbf{x},t)\right)+F_\alpha\Delta t, \quad (1)$$

$$g_\alpha(\mathbf{x}+\mathbf{e}_\alpha\Delta t, t+\Delta t)-g_\alpha(\mathbf{x},t) = -\frac{1}{\tau_g}\left(g_\alpha(\mathbf{x},t)-g_\alpha^{eq}(\mathbf{x},t)\right)+G_\alpha\Delta t, \quad (2)$$

where $f_\alpha(\mathbf{x}, t)$, $g_\alpha(\mathbf{x}, t)$ denote the density and temperature distribution functions for the discrete velocity $\mathbf{e}_\alpha$, Δt is the time step, and τ_f, τ_g are the dimensionless relaxation times of flow and thermal fields, respectively. F_α, G_α are discrete force and heat source/sink terms which are defined in the following section.

In the lattice models, the local equilibrium density and temperature distribution functions $f_\alpha{}^{eq}(\mathbf{x}, t)$, $g_\alpha{}^{eq}(\mathbf{x}, t)$ are given by

$$f_\alpha^{eq}(\mathbf{x},t) = \omega_\alpha\rho\left(1+\frac{\mathbf{e}_\alpha\cdot\mathbf{u}}{c_s^2}+\frac{(\mathbf{e}_\alpha\cdot\mathbf{u})^2}{2c_s^4}-\frac{\mathbf{u}^2}{2c_s^2}\right), \quad (3)$$

$$g_\alpha^{eq}(\mathbf{x},t) = \omega_\alpha T\left(1+\frac{\mathbf{e}_\alpha\cdot\mathbf{u}}{c_s^2}+\frac{(\mathbf{e}_\alpha\cdot\mathbf{u})^2}{2c_s^4}-\frac{\mathbf{u}^2}{2c_s^2}\right), \quad (4)$$

where $\mathbf{u}$ is the fluid velocity, c_s^2 is the lattice sound speed, and ω_α is the weight coefficient which depends on the lattice velocity model.

The D2Q9 model is applied in this study, and the discrete velocity set is defined as

$$\mathbf{e}_\alpha = \begin{cases} (0,0), & \alpha = 0 \\ \left(\cos\left[(\alpha-1)\frac{\pi}{2}\right], \sin\left[(\alpha-1)\frac{\pi}{2}\right]\right)c, & \alpha = 1,2,3,4 \\ \sqrt{2}\left(\cos\left[(2\alpha-1)\frac{\pi}{4}\right], \sin\left[(2\alpha-1)\frac{\pi}{4}\right]\right)c, & \alpha = 5,6,7,8, \end{cases} \quad (5)$$

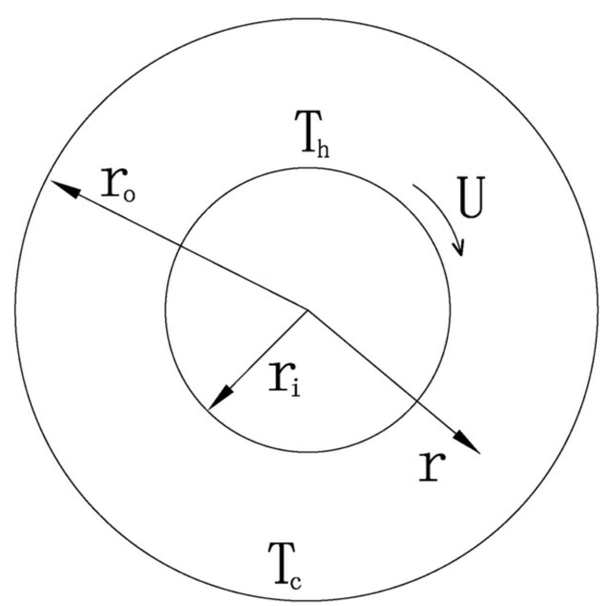

Fig. 2. The schematic of the rotation flow with heat transfer.