International Journal of Heat and Mass Transfer 126 (2018) 1059–1070

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

An immersed boundary-discrete unified gas kinetic scheme for simulating natural convection involving curved surfaces

Chao Li [a], Lian-Ping Wang [b,c,*]

[a] Marine Engineering College, Dalian Maritime University, Dalian 116026, Liaoning, China
[b] Department of Mechanical Engineering, University of Delaware, Newark, DE 19716, USA
[c] Department of Mechanics and Aerospace Engineering, Southern University of Science and Technology, Shenzhen 518055, Guangdong, China

ARTICLE INFO

Article history:
Received 7 January 2018
Received in revised form 31 March 2018
Accepted 28 April 2018

Keywords:
Natural convection
IB-DUGKS
Strang-splitting
Curved boundary

ABSTRACT

In this work, an immersed boundary-discrete unified gas kinetic scheme (IB-DUGKS) is proposed and presented for the simulation of natural convection with a curved body surface. In this method, two distribution functions are employed for velocity and temperature field, respectively, and they are coupled under the Boussinesq approximation. The IB-DUGKS provides an effective way for the DUGKS to treat a curved boundary. The Strang-splitting method is used to handle the IB force, and its accuracy is first validated by comparing with another implementation method for the base case of natural convection in a square cavity. The widely used direct-forcing immersed boundary method is adopted due to its simplicity, with an iteration procedure to ensure the accuracy of no-slip condition on the immersed boundary. Natural convection between an outer square and an inner circular cylinder is then simulated under different geometric configurations, including different aspect ratios and locations of the cylinder relative to the cavity. The numerical results are in excellent agreement with the results from the literature, confirming the accuracy and robustness of the proposed method.

1. Introduction

Natural convection between a body and an enclosure has received a great deal of attention in the past decades, as it is relevant to many industrial applications such as heat exchangers, cooling of electronic equipment, and thermal storage systems [1]. In this paper, the specific geometric configuration of interest is a cold outer square enclosure and a hot inner circular cylinder. The natural convection problem of this geometric configuration has also been investigated by other people in recent years, such as Moukalled and Acharya [2] and Shu and Zhu [3], and has often served as a benchmark case to verify new numerical methods [4–6]. In the present study, an immersed boundary-discrete unified gas kinetic scheme (IB-DUGKS) is developed to investigate such a natural convection problem involving a curved surface.

IB-DUGKS is a kinetic method solving a model Boltzmann equation. Unlike the traditional CFD methods which are based on solving the Navier–Stokes equations, the kinetic methods are based on the kinetic theory. The kinetic methods provide a connection between the macroscopic hydrodynamics and the microscopic physics, and are sometime referred to as mesoscopic methods. Among the different kinds of kinetic methods, the gas kinetic scheme (GKS) [7] and lattice Boltzmann method (LBM) [8] are widely used and have been developed rapidly in recent years. Based on GKS, a unified GKS (UGKS) for all Knudsen number flows was developed by Xu and Huang [9]. And recently, the DUGKS was developed by Guo et al. which combines the advantages of UGKS and LBM [10,11]. It is a finite volume method and derived directly from the Boltzmann equation. Compared with LBM, the DUGKS is more flexible in application, such as fully decoupled time and space steps, and also a non-uniform mesh can be easily employed.

Since DUGKS is relatively new, only a few studies have emerged to explore the potential applications of DUGKS. Wang et al. [12] proposed a coupled DUGKS for Boussinesq flows and the Rayleigh-Bénard convection and natural convection in a square cavity were investigated. Wu et al. [13] proposed a general method to allow the DUGKS to handle an external force term by adding the force term into the Boltzmann equation and DUGKS procedure. Zhu et al. [14] successfully extended the DUGKS to unstructured meshes. Guo and Xu [15] extended the DUGKS to simulate the whole multiscale heat transfer process based on the phonon Boltzmann transport equation. Bo et al. [16] investigated 3D Taylor–Green vortex flow and turbulent channel flow using DUGKS. Zhu et al. [17] developed an open source OpenFOAM solver for the

* Corresponding author at: Department of Mechanical Engineering, University of Delaware, Newark, DE 19716, USA.

E-mail addresses: lichao2013@dlmu.edu.cn (C. Li), lwang@udel.edu (L.-P. Wang).

https://doi.org/10.1016/j.ijheatmasstransfer.2018.04.166
0017-9310/

Boltzmann model equation with DUGKS. Wang et al. [18] conducted a systematic numerical study of three-dimensional natural convection in a differentially heated cubical cavity with Rayleigh number up to 10^{10}. Recently, the IB-DUGKS has been developed for isothermal flows with curved boundary by Tao et al. [19]. So far, DUGKS has not been applied to heat transfer problems with a curved boundary.

To incorporate curved boundaries, we shall consider the immersed boundary method (IBM) which was first proposed by Peskin in the early 1970s [20]. Due to its simplicity in implementation and flexibility in application, it has drawn particular attention in the recent decades [21–23]. The main idea of IBM is to use two set of grids for the simulation, with a fixed Eulerian grid covering the whole domain for the fluid, the Lagrangian points representing the immersed boundary. The interaction between the fluid and the immersed boundary is handled through the IB force. IBM was first used to simulate elastic material boundary, and the IB force on the boundary points can be determined by the deformation under Hooke's law [20,21]. When the force is distributed to the fluid through a smooth delta function, the effect of the real boundary can be approximated by the IB force of the immersed boundary. The original method to calculate the IB force can be called the penalty or feedback forcing method. Another popular way to decide the IB force is the direct forcing method proposed by Mohd-Yusof [22]. It is simpler to implement. But the original direct forcing method cannot ensure the no-slip condition on the boundary due to the delta function interpolation errors causing the streamlines to penetrate through the immersed boundary. To avoid this problem, the multi-direct forcing method and the implicit direct forcing method were developed. The multi-direct forcing method was first used by Luo et al. [24], and the details will be described in Section 2.2. The implicit direct forcing method was first proposed by Wu et al. [25]. In this implicit method, one does not calculate the IB force. Instead, the velocity corrections at all boundary points are considered as unknowns which are computed in such a way that the non-slip boundary condition at the boundary points is enforced. The drawback of the implicit direct forcing method is that one need to solve a matrix system, but the no-slip boundary condition can be satisfied precisely. Besides these two main methods (feedback forcing and direct forcing methods), there are many other ways to implement the IBM. One of them is the interpolation-based scheme proposed by Kim et al. [26,27], which is based on a finite volume approach on a staggered mesh together with a fractional-step method. The momentum forcing and the mass source/sink are applied on the body surface or inside the body to satisfy the no-slip boundary condition on the immersed boundary and the continuity for the cell containing the immersed boundary, respectively. The heat source/sink is introduced on the body surface or inside the body to satisfy the isothermal or *iso*-heat-flux condition on the immersed boundary. A second-order linear or bilinear interpolation scheme is used to satisfy the no-slip velocity on the immersed boundary, which is numerically stable regardless of the relative position between the grid and the immersed boundary. Kim et al. had validated their method with isothermal flow and heat transfer problems [26,27], which also showed the capability of their method. One can find other versions of the IBM from the literature or from Refs. [28–30].

Within the conventional CFD which solves the Navier-Stokes equations directly, IBM is well established for isothermal problems. A few non-isothermal studies using the IBM are noted here. Kim et al. [4] investigated natural convection between a cold outer square and a hot inner circular cylinder with the interpolation-based IBM. Jiong et al. [6] investigated natural convection in a square enclosure with feedback forcing IBM. Wang et al. [31] investigated natural and forced convection problems with the direct forcing IBM. These and other studies [32–35] reveal that IBM is a competent method for solving a thermal flow within conventional CFD.

The aim of present work is to combine DUGKS (a mesoscopic flow solver) and IBM (a curved boundary treatment) in order to formulate a mesoscopic simulation tool for natural convection problems with complex geometries. The rest of this paper is organized as follows. In Section 2, a brief introduction of DUGKS and IBM, as well as how to couple the two methods are described. In Section 3, the accuracy of the present method is validated by comparing the simulation results for several benchmark problems with the data from the literature. Finally, a brief summary and conclusions are presented in Section 4.

2. Simulation method

In this section, the DUGKS algorithm is described first. Then the direct-forcing immersed boundary method is introduced. At last two different ways for the DUGKS to incorporate an external force term are given.

2.1. Discrete unified gas kinetic scheme

DUGKS was first proposed by Guo et al. [10], one can also find the details about this method from the previous studies [12,13]. Here a brief introduction of the method is given.

2.1.1. DUGKS for velocity field

DUGKS begins with the Boltzmann equation with the BGK collision model [10]

$$\frac{\partial f}{\partial t} + \xi \cdot \nabla f = \Omega \equiv \frac{f^{eq} - f}{\tau_v}, \tag{1}$$

where f is the distribution function for the velocity field, $f = f(x, \xi, t)$ with space $\boldsymbol{x}$, time t and velocity ξ. Ω is the collision term, τ_v is the relaxation time and related to the viscosity coefficient. f^{eq} is the Maxwellian equilibrium state and has the following form:

$$f^{eq} = \frac{\rho}{(2\pi RT_1)^{D/2}} \exp\left(-\frac{(\xi - \mathbf{u})^2}{2RT_1}\right), \tag{2}$$

where ρ is density of the fluid, R is the gas constant, T_1 is a constant temperature, $\mathbf{u}$ is the macroscopic velocity of the fluid, D is the spatial dimension. Here $RT_1 = c_s^2$, c_s is the artificial sound speed. The hydrodynamic variables can be obtained as:

$$\rho = \int f d\xi, \quad \rho\mathbf{u} = \int \xi f d\xi. \tag{3}$$

The DUGKS is a finite volume method, and the flow domain can be divided into a set of control volumes V_j which are centered at $\boldsymbol{x_j}$. Integrating Eq. (1) on V_j from time t_n to t_{n+1}, and using the midpoint rule for the integration of the convection term and trapezoidal rule for the collision term, one can obtain

$$f_j^{n+1} - f_j^n + \frac{\Delta t}{|V_j|} F^{n+1/2} = \frac{\Delta t}{2} (\Omega_j^{n+1} + \Omega_j^n), \tag{4}$$

where Δt is the time step, and

$$F^{n+1/2} = \int_{\partial V_j} (\xi \cdot \mathbf{n}) f(x, t_{n+1/2}) dS \tag{5}$$

is the microflux across the interface, $\boldsymbol{n}$ is the unit vector normal to the cell interface. But Eq. (4) used to update the distribution function f is implicit, so two new distribution functions are defined

$$\tilde{f}_j = f_j - \frac{\Delta t}{2} \Omega_j, \quad \tilde{f}_j^+ = f_j + \frac{\Delta t}{2} \Omega_j. \tag{6}$$