International Journal of Thermal Sciences 82 (2014) 23–33

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

Three dimensional thermal Lattice Boltzmann simulation of heating/cooling spheres falling in a Newtonian liquid

Zahra Hashemi, Omid Abouali*, Reza Kamali

School of Mechanical Engineering, Shiraz University, Shiraz, Iran

ARTICLE INFO

Article history:
Received 14 May 2013
Received in revised form
4 February 2014
Accepted 9 March 2014
Available online

Keywords:
Lattice Boltzmann method
Heat transfer
Fluid-particle interaction
Falling particle
Three-dimensional numerical simulation

ABSTRACT

This paper presents a numerical study of heat transfer from spheres settling under gravity in a box filled with liquid. Three-dimensional Lattice Boltzmann Method is applied to simulate fluid-particle interaction. Firstly the developed numerical model is validated in comparison with experimental and numerical data for a falling sphere in a box filled with liquid. Then the effects of Reynolds, Prandtl and Grashof numbers (*Re*, *Pr*, *Gr*) are investigated for a settling particle at fixed/varied temperature. The time variations of velocity, height and Nusselt number of the settling particle are also investigated. The results depict that the maximum settling velocity of the particle at varied surface temperature is higher at lower Reynolds numbers. As the Reynolds number increases, the settling velocity of the particle for both cases of constant and varied temperature is the same. However, the Nusselt number of the particle at varied temperature is lower compared with that for the particle at constant surface temperature. Increasing the Grashof number leads to slower settling and larger average Nusselt number. Finally sedimentation of 30 hot spherical particles in an enclosure and their hydraulic and heat transfer interactions with the surrounding fluid are studied. The results depict how the presence of heat transfer phenomena can significantly alter the behavior of settling particles.

1. Introduction

Sedimentation of solid particles in a viscous fluid can be found in many natural and biological situations as well as in industrial applications, such as the sedimentation of sand particles in rivers, the flow of blood cells in capillaries, fuel spray, separating particles with different sizes or densities [1]. In many cases, the heat transfer and momentum interaction between particles and fluid are of interest to investigate. Over the years, fluid and stationary particle interactions in non-isothermal situations have been studied extensively [2–8]. These studies include different methods to simulate particles with different shapes in two or three-dimension. In addition, in the literature there are a wide variety of studies regarding to sedimentation of a single particle in the fluid. A review of the early works can be found in Clift et al. [9] Chhabra [10] and McKinley [11]. Gan et al. [12] studied thermal interactions between particle and surrounding fluid, using two dimensional Arbitrary Lagrangian–Eulerian (ALE) finite-element method (FEM). They showed that thermal convection may fundamentally change the way that particles move and interact. Since the ALE/FEM is a body-fitted method, a body-fitted mesh is needed and re-meshing is usually required when the particles move. The ALE method is very computationally expensive; especially for three-dimensional particulate flow that the re-meshing procedure would be very difficult and time-consuming. Yu et al. [13] applied the fictitious domain method to investigate two-dimensional particulate flow with convection heat transfer for various temperature boundary conditions on the particles. Mixed convection around a hot spherical particle settling in a quiescent fluid was studied by Bhattacharyya and Singh [14]. The authors solved the vorticity transport and energy equations to analyze the heat transfer phenomenon. Feng and Michaelides [15] developed a numerical method to solve momentum and energy equations in two-dimensional particulate flows by using direct numerical simulation (DNS). They computed the effect of thermal interaction on vertical velocity of the particle and also simulated sedimentation of 56 hot circular particles in a two-dimensional enclosure. Dan and Wachs [16] used distributed Lagrange multiplier/fictitious domain (DLM/FD) method to handle three-dimensional problems and they simulated the sedimentation of a single spherical particle with heat transfer in a semi-infinite channel. Their study was based on the assumption of a constant particle temperature. Wachs [17] investigated the problem of

* Corresponding author.
E-mail addresses: abouali@shirazu.ac.ir, aboualio@yahoo.com (O. Abouali).

http://dx.doi.org/10.1016/j.ijthermalsci.2014.03.008
1290-0729/

spherical catalyst particles rising in an enclosure due to the natural convection phenomenon using DLM/FD method. Shao et al. [18] combined fictitious domain and sharp interface methods to handle particle motion with heat transfer effects. Some two and three-dimensional test cases were simulated to valid the method. Although the fictitious domain methods use a non-body-fitted mesh and do not require re-meshing, however in these methods the accuracy of boundary approximations is low, leading to a great limitation in using such methods for solving three dimensional problems containing fluid–particles interaction [19]. Deen et al. [20] developed a DNS method to simulate flow field and heat transfer in dense fluid-particle systems. Ström and Sasic [21] proposed a multiphase DNS method based on the VOF (volume of fluid) technique to simulate non-isothermal moving particles. They showed that their proposed method is applicable for two- and three-dimensional problems.

Particulate flow systems are moving boundary problems. In the field of sedimentation problems, most of the conventional CFD methods based on the finite element/volume methods need to generate new, geometrically adapted grids. This process in particulate flows with large number of particles can be complicated, especially in three-dimensional cases. In 1994, the Lattice Boltzmann method (LBM) was successfully applied to simulate fluid–particle interactions [22,23]. The results proved that LBM is an alternative and efficient option for simulating particulate suspensions [24–28]. The LBM provides a robust numerical scheme that can efficiently treat the complex system of moving particles. A further advantage of this method is that it can be parallelized at high computational efficiency. It should be pointed out that, like the Immersed Boundary [15] and Fictitious Domain [13] techniques, the Lattice Boltzmann method is a non-body-fitted method which employs a fixed mesh.

The objective of this paper is to use the three-dimensional thermal LBM to investigate heat transfer from spheres settling under gravity in a box filled with liquid. The three-dimensional thermal LBM is combined with the equations of motion of the particles according to Newtonian dynamic to study the fluid–particles interactions.

The earlier developed 2D TLBM computer code [29] is extended to handle the three-dimensional flow fields with fluid-particle interaction. The momentum-exchange method is used to evaluate the hydrodynamic forces acting on the particle. The effects of Reynolds, Prandtl and Grashof numbers are investigated for a settling particle at fixed/varied temperatures. In addition, the sedimentation of 30 hot spherical particles in a three-dimensional enclosure is modeled.

2. Problem definition

In order to study the heat transfer from spherical particles settling under gravity, a three-dimensional vertical channel is used. A grid study is performed and a channel with height of 128 lattice units (lu) and cross section of 80 × 80 lu, as sketched in Fig. 1, is chosen. At $t = 0$, hot spherical particles with fixed diameter of 14 lu and different density ratios of 2.75, 4.1 and 5.48 are set in the middle of the channel, 96 lu above from the bottom. The particle is initially at rest in a fluid that fills the vertical channel. The solid particles are assumed to have high thermal conductivity and, as a result, uniform temperature in each instant.

Two different cases of settling particle at fixed and varied (time-dependent) temperatures are investigated. The no-slip boundary condition is used for channel walls and particle surface. For thermal part of the problem the constant temperature boundary condition is applied to channel walls and particle surface as well. The Boussinesq approximation is employed for the coupling between the

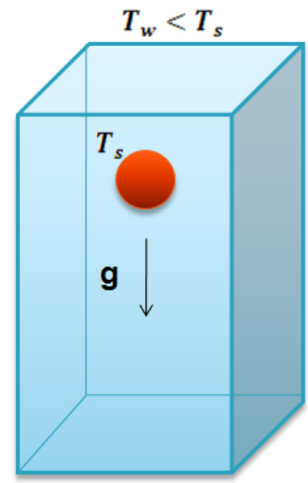

Fig. 1. A schematic of a spherical particle placed in a box.

temperature and flow fields and studying the momentum and heat interactions between the particles and the fluid.

3. Numerical implementation

3.1. Simulation of surrounding flow field by TLBM

The LBM, which was proposed in 1980s [27,30,31], is a particle-based numerical method for flow field simulations. In this method, the macroscopic variables such as velocity, temperature and pressure are not solved directly and a mesoscopic simulation model is used. To apply this method, the fluid domain is discretized in regular square cells or lattices. Each lattice is connected to its neighbors by some links. For D_3Q_{19} model, Fig. 2, there are eighteen links and fluid particles stream only on these links with eighteen different velocities:

$$e_\alpha = \begin{cases} (0,0,0) & \alpha = 0 \\ (\pm 1,0,0)c, (0,\pm 1,0)c, (0,0,\pm 1)c & \alpha = 1\sim 6 \\ (\pm 1,\pm 1,0)c, (0,\pm 1,\pm 1)c, (\pm 1,0,\pm 1)c & \alpha = 7\sim 18 \end{cases} \tag{1}$$

where c is the lattice speed and determined by $\delta x/\delta t$ and e_0 shows the moving particles with zero velocity, which remain at the node.

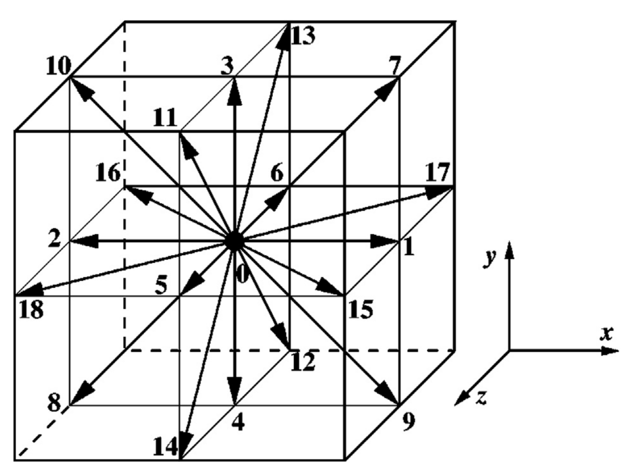

Fig. 2. LBM D_3Q_{19} model.