International Journal of Heat and Mass Transfer 65 (2013) 235–243

Contents lists available at SciVerse ScienceDirect

International Journal of Heat and Mass Transfer

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

Thermal three-dimensional Lattice Boltzmann simulations of suspended solid particles in microchannels

Zahra Hashemi, Omid Abouali *, Reza Kamali

School of Mechanical Engineering, Shiraz University, Shiraz, Iran

ARTICLE INFO

Article history:
Received 28 March 2013
Received in revised form 23 May 2013
Accepted 3 June 2013
Available online 2 July 2013

Keywords:
Fluid–particle interaction
Thermal Lattice Boltzmann method (TLBM)
Microchannel
3D numerical simulation

ABSTRACT

This paper presents a three-dimensional thermal Lattice Boltzmann model to simulate the interaction of suspended solid particles with the flow field and their effects on hydrodynamic and thermal performance of microchannels. A 19-bit single-relaxation-time Lattice Boltzmann method (D3Q19) is used to perform thermal fluid flow, while the Newtonian dynamic equations are solved to investigate the transport of the suspended solid particles. The needed forces in equations of the particle motion are evaluated by the momentum exchange method. The effects of solid spherical particles with various diameters on the fluid flow and heat transfer enhancement in a rectangular microchannel, at different Reynolds numbers, are investigated and discussed. Three cases of stationary, constant velocity and freely moving single particle are investigated which the results show the latter case has the highest heat transfer enhancement and the lowest pressure drop. The particulate flows in the microchannel for both cases of 15 stationary and freely moving particles are also studied and the hydraulic and thermal performances are compared.

1. Introduction

Particulate suspensions can be found in many industrial and engineering applications such as aerospace, chemical, geomechanical, environmental fields and thermal systems such as heat exchangers. Interaction of suspended solid particles and flowing fluid creates some changes in the flow and thermal field. In many engineering applications, such as heat exchangers, moving machines and fluid machineries, the hydrodynamic and thermal performance of systems should be investigated including the variations of both flow and thermal fields due to the moving objects. So, examining the physical behavior of suspended particles in these systems has been attracting attention of the researchers.

In fluid–particle interaction, the size of the particle relative to the domain size is an important factor. When the particle sizes are comparable with the domain size and cover more than one computational grid, for example in microchannels, the presence of particles can greatly affect the flow field pattern and consequently, the interaction of solid particle with flow has to be considered. Due to the complexity of the interacting particles and lack of powerful analytical tools, numerical simulation is the best way to efficiently predict this phenomenon with lower cost.

Dhole et al. [1] investigated the forced convection heat transfer from an unconfined sphere by using a finite volume method. Effects of fluid flow and heat transfer from a circular cylinder located between parallel plates are investigated by Khan et al. [2] using an integral approach. Mettu et al. [3] studied heat transfer from a confined circular cylinder in a plane channel numerically. Ahmadi Motlagh and Hashemabadi [4] modeled heat transfer from circular cylindrical particles. Maheshwari et al. [5] reported numerical results for heat transfer from a single sphere and an array of three spheres. Chatterjee and Mondal [6] performed a simulation for a stationary two-dimensional square cylinder and investigated the effect of cross buoyancy on vortex shedding. Gandikota et al. [7] studied the effect of thermal buoyancy on heat transfer around a circular cylinder. They employed a two dimensional finite volume model for their analysis. All the researches mentioned above and other works done in the field of fluid–particle thermal interactions [8–10] have focused on the stationary objects.

In order to simulate moving particles in particulate flows numerically, three common approaches have been widely used [11]: two-phase continuum model, discrete particle model and Direct Numerical Simulation (DNS). The first technique uses two momentum equations for solid and liquid phases and the apparent viscosity and drag coefficient used in these equations can deal the fluid–particle interactions. In second model the particles are treated as the points and empirical relations are used to evaluate the forces acting on particle. In each time step the equation of motion is solved to update the position of particles. Recently due to the rapid improvement of computer power, the Direct Numerical Simulation based on the Navier–Stokes equations or discrete Lattice-Boltzmann equation has become popular as a practical

---

* Corresponding author. Tel.: +98 7116133034.
E-mail address: abouali@shirazu.ac.ir (O. Abouali).

0017-9310/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.ijheatmasstransfer.2013.06.005

and important tool to tackle particulate flows [12]. This method is classified in two approaches [12,13]: the boundary-fitted methods and non-boundary-fitted methods. The boundary-fitted approach such as the arbitrary Lagrangian–Eulerian (ALE) finite element method [14,15], is very complex since the fluid flow is computed on a boundary-fitted mesh and re-meshing is required when the particles are moving. However, non-boundary-fitted method, such as the Lattice Boltzmann Method [16] and fictitious domain method [17], do not require re-meshing and stationary grid is used for the fluid flow. Generally for simulation of particulate flows, the non-boundary-fitted methods are simpler and more efficient than the boundary-fitted methods [11–13].

In 1994, the Lattice Boltzmann method (LBM) was successfully applied to simulate fluid–particle interactions [18,19]. LBM has been used to simulate a variety of fluid phenomena. The results proved that LBM is an alternative and efficient option for simulating particulate suspensions. One significant advantage of LBM is its computational efficiency as it is a straightforward method and easy to implement on parallel processors since a solution of any linear system of equations is not required.

Some limited studies of moving objects in a thermal fluid flow based on finite element methods and direct numerical simulations (DNS) are available in literature. Zhao et al. [20] presented finite elements based techniques for simulation of flow-particle interactions including forced or natural convection in two dimensions. The effect of particles on temperature distribution in turbulent flow was studied both experimentally and numerically (DNS) by Hetsroni et al. [21].Yang and Lai [22] used two-dimensional LBM to simulate forced convection flow of nanofluids in a microchannel and studied the effects of small nanoparticles on heat transfer rate. Chatterjee [23] introduced an enthalpy-based thermal Lattice Boltzmann model and simulated the thermo-fluidic transport problems. Khiabani et al. [24] investigated the effect of fixed and suspended cylindrical solid particles on heat transfer in a two-dimensional channel flow using LBM. They used 2-D Lattice Boltzmann equations only for fluid flow and for thermal transport the conventional form of energy equation was employed.

In the literature for the particulate flows only the hydrodynamic interactions have been studied by LBM so far and if the thermal interaction is studied, like Ref. [24], the temperature field has been found using conventional energy equation, not the thermal Lattice Boltzmann method. Moreover, in such cases two-dimensional models have been used to simulate the thermal fluid–particle interactions.

In present paper, the three-dimensional thermal LBM together with the Newtonian dynamic equations are used to investigate the effect of solid spherical particles on flow field pattern and temperature distribution in a microchannel. The earlier developed 2D TLBM computer code [25] is modified to handle the 3D flow fields with interaction of fluid and particle. The momentum-exchange method is used to evaluate the hydrodynamic forces exerted on the particle. The effects of Reynolds number and blockage ratio on microchannel performance are investigated for three cases: stationary, moving with constant velocity, and freely suspended particles in flow.

2. Problem definition

In order to study the effects of suspended solid particles on hydrodynamic and thermal performance of a microchannel, spherical particles with diameter d and density ratio of 1.02 were considered in a microchannel with square cross-sectional area of $H^2 = 50 \times 50\ \mu m^2$ and length of $L = 220\ \mu m$. A schematic of the physical domains and related conditions for microchannels containing one and multiple spherical particles are presented in Fig. 1. The computations are performed for Pr = 7. Three different cases of stationary, moving with constant velocity and freely moving particles in a microchannel were investigated. Since the Knudsen number for liquid flow in microchannels is much less than 0.01, the no-slip boundary condition is applied for microchannel walls [26] and particle surface. At the inlet, a parabolic velocity profile is considered for forced flow and velocity gradient normal to the outlet boundary is set to zero. For thermal part of the problem, constant temperature conditions T_w are considered for the channel walls, and the particle is assumed to be adiabatic. The temperature of the flow entering the channel is assumed uniform and at the outlet the temperature gradient normal to the outlet boundary is set to zero. For thermal boundary conditions of the channel walls, the idea of D'Orazio and Succi [27] is extended for three-dimensional simulation. The needed hydrodynamic and internal energy distribution functions for particle surface will be presented in Section 3.1.

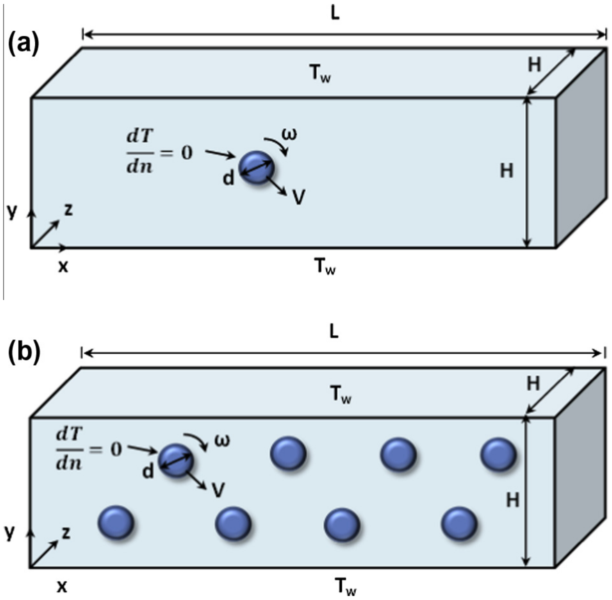

Fig. 1. Schematic of the physical domain.

Here two dimensionless numbers are used to compare the numerical results. The Reynolds number that is based on the average inlet velocity and channel width, $Re = UH/\upsilon$ and the Blockage ratio b, which is defined as the ratio of cross section area of particle to that for the channel, $b = (\pi d^2/4)/H^2$.

In order to compare the thermal performance of the microchannel, the local Nusselt number is used which can be defined as

$$Nu_x = \frac{D_h}{\Delta T}\left(-\frac{\partial T}{\partial n}\right)_w \quad (1)$$

where D_h is the hydraulic diameter and n is the normal direction to the wall. The average Nusselt number along the channel wall is given by

$$\overline{Nu} = \frac{1}{L}\int_0^L Nu_x dx \quad (2)$$

3. Numerical method

The LBM is a new computational technique that has been achieved great success for simulation of fluid dynamic problems. In the LBM, the fluid motion is simulated by particle movement and collision on a uniform lattice. It uses a particle distribution