Powder Technology 262 (2014) 62–70

Contents lists available at ScienceDirect

Powder Technology

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

Direct numerical simulation of heat and mass transfer of spheres in a fluidized bed

Zhi-Gang Feng *, Samuel Gem Musong

Department of Mechanical Engineering, University of Texas at San Antonio, TX, United States

ARTICLE INFO

Article history:
Received 6 December 2013
Received in revised form 17 February 2014
Accepted 4 April 2014
Available online 24 April 2014

Keywords:
Fluidized bed
Heat and mass transfer
Forced convection
Direct numerical simulation
Resolved discrete particle method
Immersed boundary method

ABSTRACT

We have developed a direct numerical simulation approach combined with the immersed boundary (DNS–IB) method for studying heat transfer in particulate flows. In this method, fluid velocity and temperature fields are obtained by solving the modified momentum and heat transfer equations, which are due to the presence of heated particles in the fluid; particles are tracked individually and their velocities and positions are solved based on the equations of linear and angular motions; particle temperature is assumed to be constant. The momentum and heat exchanges between a particle and the surrounding fluid at its surface are resolved using the immersed boundary method with the direct forcing scheme. The DNS–IB method has been used to study the heat transfer of 225 heated spheres in a fluidized bed. By exploring the rich data generated from the DNS–IB simulations, we are able to obtain statistically averaged fluid and particle velocities as well as the overall heat transfer rate in the fluidized bed. Good agreement between the current study and the one by Pan et al. (2002) is found for the hydrodynamic properties of the bed such as pressure gradients within the bed and the relationship between fluidization velocity and bed solid fraction. The particle-averaged Nusselt number is found to increase as the fluidization velocity increases and the bed height rises; particles at the entrance of the bed tend to have the maximum heat transfer rate because of the higher particle–fluid temperature gradients in this region; as the fluid moves upward in the bed, it gets warmer, which reduces particle–fluid temperature gradients and decreases the transfer rate of particles.

Published by Elsevier B.V.

1. Introduction

Fluidization involves a fluid flow that is supplied through a bed of solid particles at a sufficient velocity such that the entire suspension of solid particles behaves like a fluid. Because of enhanced particle–fluid heat transfer and chemical reaction rates in fluidized beds, fluidization is used in several industrial applications such as chemical reactors, coal combustion, fluid catalytic cracking, gasification, coating processes, and pyrolysis. Fluidized bed combustion, for example, offers several advantages over conventional combustion technologies such as better heat transfer characteristics due to uniform particle mixing, lower temperature requirements, near isothermal process conditions, and continuous operation ability. To better understand the flow dynamics and heat transfer of particulate flows, numerical simulation could provide an efficient and accurate technique in predicting key operating parameters such as pressure drop, minimum fluidization velocity, solid fraction, and heat transfer coefficient without any prior testing.

Most of the early studies on the heat transfer in fluidized beds lead to physical or mechanistic models based on experimental measurements. One of the first studies was done by Mickley and Fairbanks [18], who studied the heat transfer mechanism between fluidized beds and the surfaces they contact. They found the heat transfer coefficient to be proportional to the square root of the thermal conductivity of the quiescent beds of different gases for the same particle constituents. Decker and Glicksman [4] proposed a heat transfer model for immersed surfaces in large particle fluidized beds (1 mm or larger), showing an increase in the heat transfer rate by gas convection with increase in the particle size. Arters and Fan [1] studied solid–liquid mass transfer in fluidized beds and proposed an axial dispersion model. They developed a correlation which is able to accurately predict the mass transfer in both two- and three-phase fluidized beds. Basu and Nag [3] developed a more realistic hydrodynamic model to predict the heat transfer in a circulating fluidized bed (CFB) by estimating the residence time from Subbarao's [25] cluster theory; the predicted that heat transfer coefficient was expressed in terms of cluster residence time, which agrees well with the previous experimental data of Kiang et al. [15] and Martin [17] for different superficial velocities and solid circulating rates. The heat transfer for different geometry immersed in a fluidized bed has also been studied experimentally by Penny et al. [21] and Baskakov et al. [2].

The challenge with the experimental values of the heat transfer coefficients in general is the low accuracy in-bed temperature measurements and oversimplifications in the flow models. In order to find an averaged heat transfer coefficient over the whole bed, one has to

* Corresponding author. Tel.: +1 210 4585737; fax: +1 210 4586504.
E-mail address: zhigang.feng@utsa.edu (Z.-G. Feng).

http://dx.doi.org/10.1016/j.powtec.2014.04.019
0032-5910/Published by Elsevier B.V.

assume a flow model for the fluid and the particles and perform gross temperature measurements on the entering stream, leaving stream, and bed. Usually, temperature measurements taken in beds with thermocouples have been interpreted very differently and direct measurements of solid particle temperatures are even more difficult and unreliable [12]. Numerical modeling can be a vital technique that is suitable to approximate the local and average heat transfer coefficient of a fluidized bed.

Three methods are commonly used to study particulate flows: the DNS or the resolved discrete particle method, the discrete element method combined with computational fluid dynamics for fluid phase (DEM–CFD) or unresolved discrete particle method, and the two-fluid model. The DNS and DEM–CFD use the Lagrangian approach to track particle dynamics while the TFM takes the Eulerian viewpoint to describe the particles, treating them as a continuum "fluid." However, the DNS differs significantly from DEM–CFD in resolving particle–fluid momentum and heat interactions. The DNS uses a fine grid to resolve the flow field surrounding each particle; it solves the drag, torque, and heat transfer rate of particles directly and produces more accurate results. On the other hand, the DEM–CFD uses a coarse grid in which a computational cell may contain multiple particles; it employs empirical correlations for particle–fluid momentum and heat transfer. In fact, simulation results from the DNS can be used to provide constitution equations or particle drag and heat transfer correlations to less detailed models such as the DEM–CFD and the TFM. Pan et al. [20] is probably the first group to study particle fluidization in a liquid–solid bed using a fictitious domain based DNS approach. They were able to recover the celebrated Richardson [24] power-law that relates the particle fluidization velocity to the fluid fraction. Other immersed boundary based DNS has also been developed to study particulate flows [6,27] after the successful introduction of the immersed boundary concept by Peskin [22] which makes the treatment of solid–fluid interaction much easier.

In the recent years, the DNS has attracted much attention in studying particulate flows with heat transfer. Yu et al. [31] extended the fictitious domain method to study heat transfer in particulate flows. Feng and Michaelides [7,8] developed a DNS combined with immersed boundary (DNS–IB) to study two-dimensional heat transfer between flow and moving particles. Kang and Hassan [14] implemented a lattice Boltzmann method based two-dimensional DNS–IB for solving heat transfer problems in particulate flows. Wachs [28] implemented a parallelized DNS–IB with a fictitious domain method to study the rising of three dimensional catalyst particles in a natural convection dominated flow. Deen et al. [5] used a three-dimensional DNS–IB to investigate the heat transfer in both stationary beds and fluidized beds while Tavassoli et al. [26] used DNS–IB to study the heat transfer in stationary random arrays of spheres maintained at constant temperature. These studies have provided an extensive validation of the DNS–IB method for solving particulate flows with heat transfer.

In the present paper, we will first briefly describe the DNS–IB method for heat transfer in particulate flows and its implementation. Then, we will employ the DNS–IB to study both the mixed convection and forced convection of a heated sphere in a large domain. Results are compared to those found in the literature. Finally, we will extend the method to study the heat transfer of 225 spheres in a fluidized bed. The results of flow hydrodynamic properties such as the pressure drop, the minimum fluidization velocity, and the solid fraction are compared with those obtained by Pan et al. [20]. The overall particle-averaged Nusselt number is also obtained, from which the total heat transfer between the fluidized particles and the carry fluid could be easily determined.

2. Description of the numerical simulation method

The immersed boundary (IB) method combined with a direct forcing scheme has been implemented for solving particle motions and fluid velocity fields of particulate flows by Feng and Michaelides [6] and by Uhlmann [27]. It has also been extended to solve two-dimensional heat transfer of moving particles in a viscous fluid by Feng and Michaelides [7,8]. In the DNS–IB for particulate flows, we use two types of grids to solve the particle and fluid interactions: one is a fixed regular background grid or Eulerian grid for the entire computational domain occupied by both fluid and particles; the other is a moving grid or Lagrangian grid for outlining the immersed solid particles. For the momentum interaction between the fluid and solid boundary, the presence of the solid boundary can be represented by a virtual boundary that is assigned a distribution force. The virtual boundary behaves the same way the solid boundary would; when fluid approaches the virtual boundary, a distributing force arises that repels the fluid and prevents it from penetrating. This distribution force depends on the incoming flows and can be conveniently computed by the direct forcing scheme [19]. This force density function enforces the no-slip boundary condition on the particle surface. With respect to the thermal interaction between the fluid and the solid boundary, a thermal distribution energy density can be introduced to make the virtual boundary satisfy the no-jump temperature boundary condition at the solid surface. Under such treatment, the whole computation domain can be regarded as if it was filled with fluid. However, in the region that is occupied by solid, a special distribution force and distribution thermal energy density are used to account for the effect of the physical boundary. For particles that have different temperatures than the fluid, the presence of this term will create a temperature gradient within the fluid, which would modify the fluid properties. For relatively small temperature differences between the particles and fluid, the Boussinesq approximation has been often used successfully for coupling energy and momentum equations.

The concept of the immersed boundary that was introduced by Peskin [22], illustrated in Fig. 1, enables one to describe the entire domain Ω, that is occupied by the fluid Ω_f and solid body Ω_s, by using the modified Navier–Stokes equations for incompressible viscous flows. In the present study, we choose the imposed fluidization velocity V as the reference velocity U_{ref}, the particle diameter d as the reference length L_{ref}, and d / V as the reference time t_{ref}. The dimensionless temperature is defined as:

$$T = \frac{T_f^* - T_{f0}^*}{T_s^* - T_{f0}^*} \tag{1}$$

where T_{f0}^* is the dimensional temperature of fluid entering the bed, T_s^* is the dimensional temperature of a sphere, and T_f^* is the dimensional fluid temperature field. We also let $\vec{u}$ be the flow dimensionless velocity, p be the dimensionless pressure, and t be the dimensionless time variable. The modified momentum equations with the Boussinesq approximation in dimensionless form may be written as follows [8]:

$$\frac{\partial \vec{u}}{\partial t} + \vec{u} \cdot \vec{\nabla} \vec{u} = -\vec{\nabla} p + \frac{1}{Re} \nabla^2 \vec{u} + \frac{Gr}{Re^2} T \hat{e}_g + \vec{f}, \quad \vec{x} \in \Omega \tag{2}$$

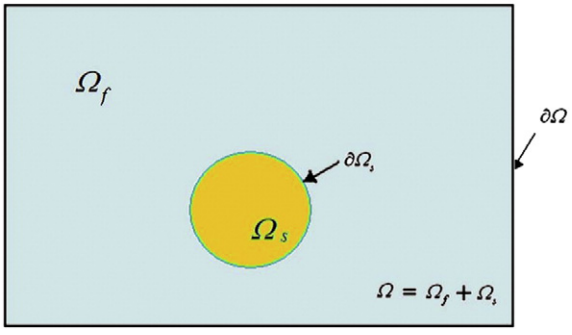

Fig. 1. Conceptual model of a circular particle suspended in a fluid.