International Journal of Heat and Mass Transfer 112 (2017) 45–60

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Fully resolved numerical simulation of interphase heat transfer in gas–solid turbulent flow

Junjie Xia [a,b], Kun Luo [a], Jianren Fan [a,*]

[a] *State Key Laboratory of Clean Energy Utilization, Zhejiang University, Hangzhou 310027, PR China*
[b] *Wuhan Second Ship Design and Research Institute, Wuhan 430200, PR China*

ARTICLE INFO

Article history:
Received 31 October 2016
Received in revised form 13 April 2017
Accepted 21 April 2017
Available online 3 May 2017

Keywords:
Multiphase flow
Heat transfer
Immersed boundary method
Nusselt number

ABSTRACT

We use a ghost-cell based high-order immersed boundary method (IBM) to study the thermal interaction between entrained solid spherical particles and a turbulent velocity- and temperature-carrying flow. The sphere diameter (D) is about eight times the Kolmogorov scale (η). The ambient turbulent field is isotropic, and the Taylor microscale Reynolds number is 50. The inflow turbulent velocity intensity varies from 0.05 to 0.1, and the intensity of temperature fluctuation varies from 0.1 to 0.4. The particle volume fractions are 0.01 and 0.02, and particle-to-fluid density ratios are 1.2, 10.0, and 100.0. It is observed that the particle-to-fluid density ratio affects the Nusselt number the most, followed by the solid volume fraction and turbulence intensity, while the effect of the intensity of temperature fluctuation is relatively small. It is also shown that correlations that have been proven to be valid for a single stationary particle can deviate significantly from the exact value obtained by directly integrating the dimensionless temperature gradient over the surface of the particle. Better estimates can only be gained by also taking the local flow and thermal conditions into consideration, in addition to the particle Reynolds number.

1. Introduction

Heat transfer between a solid dispersed phase and a fluid continuous phase is of theoretical and application importance and has attracted continued attention. Many researchers have derived correlations of Nusselt number for heat transfer from a single sphere under steady and uniform flow, from experiments [1,2] or from numerical simulation [3]. Bagchi et al. [4] showed the local Nusselt number distributions over a spherical surface at numerous Reynolds numbers. Richter and Nikrityuk [5] simulated heat transfer from spherical, cuboidal, and ellipsoidal particles, as well as a sphere with cylindrical bore [6]. All these studies are confined to a regular-shaped stationary particle subject to a steady and uniform ambient flow.

In the natural environment and in many industrial applications, millions of particles are usually dispersed in the carrier fluid, and the particles are transported by the action of hydrodynamic forces. On the other hand, particles may serve as sources of disturbances to stimulate turbulence in the multiphase flow, especially when the Reynolds number is sufficiently high. Although there are many studies on turbulence modulation by fully resolved particles [7–9], or on the effect of turbulent fluctuations on the drag and lift forces on a particle [10], studies on particle–turbulence thermal interactions are rare.

Bagchi and Kottam [11] looked into the role of freestream velocity and temperature fluctuations in modifying the mean and time-dependent heat transfer from a sphere. Their results confirmed that the generally available correlations for a steady and uniform ambient flow were also validated for predicting the instantaneous Nusselt number of a single stationary particle under turbulent ambient condition. Hashemi et al. [12] considered stationary, constant velocity and freely moving single particle transferring heat with fluid in a rectangular microchannel. But the inflow is uniform.

For multiparticle systems, Maheshwari et al. [13] investigated the effect of blockage ratio on the steady flow and heat transfer characteristics of incompressible fluid over an inline array of three spheres placed at the axis of a tube. Kao et al. [14] investigated the sphere blockage ratio on the thermal–hydraulic characteristics of a pebble with 14 spheres. Both these studies used stationary sphere arrays.

Tenneti et al. [15] designed a so-called "thermally fully developed flow" to examine the regime of validity of statistical homogeneity in the average fluid temperature field, which is the implicit assumption in two-fluid computational fluid dynamics (CFD) models. They assumed that there is no relative slip velocity between particles, so that they could average over different imple-

* Corresponding author.
E-mail address: fanjr@zju.edu.cn (J. Fan).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2017.04.103
0017-9310/

mentations with different fixed particle configurations. However, the validation of this assumption itself is questionable.

In the current article, we use a ghost-cell based high-order immersed boundary method (IBM) to simulate the heat transfer process of hundreds of freely moving spherical particles under conditions of velocity and temperature fluctuations. The main objective of this study is to investigate the influence of turbulent fluctuations on the heat transfer process between the particles and the carrier fluid. Detailed analysis of the numerical results indicates that both the particle motion and turbulent fluctuations contribute to the increase of the interphase heat transfer coefficient.

The next section (Section 2) introduces the numerical strategies to solve the governing equations of the fluid and solid phases, as well as the main feature of the IBM. Other crucial simulation details, such as turbulence generation and parameter settings, are provided in Section 3. Section 4 contains the main body of this article, with results and discussion. And finally, some concluding remarks are given in Section 5.

2. Numerical strategy

2.1. Governing equations

For constant-property viscous incompressible Newtonian fluid, the transport phenomena are governed by the conservation equations for mass, momentum, and thermal energy, in dimensionless form, given by the following expressions:

$$\nabla \cdot \mathbf{u}^* = 0, \tag{1}$$

$$\frac{\partial \mathbf{u}^*}{\partial t} + \mathbf{u}^* \cdot \nabla \mathbf{u}^* = -\nabla P^* + \frac{1}{Re}\nabla^2 \mathbf{u}^*, \tag{2}$$

$$\frac{\partial T^*}{\partial t} + \mathbf{u}^* \cdot \nabla T^* = \frac{1}{Pe}\nabla^2 T^*, \tag{3}$$

where $\mathbf{u}^*$ is the dimensionless velocity vector and P^* is the dimensionless pressure. The dimensionless temperature is defined as $T^* = (T - T_0)/(T_S - T_0)$, where T_0 is the constant far-field temperature, T_S is the isothermal particle temperature, and T is the dimensional fluid temperature. The three dimensionless characteristic numbers in the governing equations are the Reynolds number $Re = (\rho_0 \cdot U \cdot D)/\mu$, the Peclet number $Pe = Re \cdot Pr$, and the Prandtl number $Pr = (c_P \cdot \mu)/k$. Here, we take the mean inflow velocity U as the characteristic velocity and the particle diameter D as the characteristic length scale. Furthermore, ρ_0, μ, c_P, and k are the fluid density, dynamic viscosity, specific heat, and coefficient of thermal conductivity, respectively. Finally, the kinematic viscosity $v = \mu/\rho_0$.

The governing equations are solved by an independently developed fractal step-based finite difference method under uniform Cartesian grids. The pressure Poisson equation, derived by applying the divergence operator to the momentum equations, replaces the continuity Eq. (1) that is satisfied indirectly through the solution of the pressure equation. The pressure Poisson equation is discretized by the second-order center-difference scheme and solved by a successive low relaxation (SLR) method until the error of mass conservation reduces to 10^{-6}. Eqs. (2) and (3) are integrated in time using a four-stage fourth-order Runge-Kutta method with the third-order Adams–Bashforth method for convection terms and Crank–Nicolson method for diffusion terms. For example, the convective term in the heat equation, $C_T = \mathbf{u}^* \cdot \nabla T^*$, can be expressed as follows:

$$C_T^{n+1} = \frac{1}{12}\left(23C_T^n - 16C_T^{n-1} + 5C_T^{n-2}\right). \tag{4}$$

The diffusion term, $D_T = -\nabla^2 T^*/Pe$, gives the expression

$$D_T^{n+1} = \frac{1}{2}(D_T^{n+1} + D_T^n) = -\frac{1}{2}\cdot\frac{1}{h^2}\cdot\frac{1}{Pe}(LT^{*(n+1)} + LT^{*(n)}), \tag{5}$$

where L represents the discretized Laplace operator and h the grid size.

The spatial derivative in the diffusion term is evaluated by a sixth-order central compact finite difference scheme, by which the first derivative (f') of a primary variable f is evaluated as follows:

$$\frac{1}{3}(f'_{i-1} + f'_i + f'_{i+1}) = \frac{14}{9}\cdot\frac{f_{i+1} - f_{i-1}}{2h} + \frac{1}{9}\cdot\frac{f_{i+2} - f_{i-2}}{4h}, \tag{6}$$

where subscript i is the spatial index. A third-order upwind compact scheme is adopted for the convective term.

For suspended solid particles, their translational and rotational motions are governed by the Newtonian equations of motion, respectively, given as follows:

$$m_p \frac{d\vec{u}_p}{dt} = m_p \vec{g} + \vec{F}_{f\to s}, \tag{7}$$

$$I_p \frac{d\vec{\omega}_p}{dt} = \vec{T}_{f\to s}, \tag{8}$$

where m_p and I_p are the mass and the moment of inertia of the particle, respectively.

And the $f \to s$ terms represent the drag and torque exerted upon the particle by the fluid. They are calculated from integrating viscous stress and pressure contribution components around the sphere surface:

$$\vec{F}_{f\to s} = \oint_S \vec{f}_{f\to s} dS = \oint_S (\mu \nabla u \cdot \vec{n} - p\,\vec{n}) dS \quad and \quad \vec{T}_{f\to s} = \oint_S (\vec{r} - \vec{r}_p) \times \vec{f}_{f\to s} dS \tag{9}$$

where $\vec{n}$ is the outward unit normal vector, $\vec{r}$ are position vectors to points at particle surface, and $\vec{r}_p$ is the position vector to the center of the sphere.

2.2. Immersed boundary method

We used a ghost-cell based high-order IBM to represent the existence of solid particles in the fluid domain. Its core idea is approximating the Taylor series expansion on a body intercept point by an Nth-order polynomial, which consists of the nearest ghost point whose flow information is to be determined and a set of adjacent fluid points on which flow information is already known.

Mathematically, in the vicinity of the immersed boundary, a generic variable ϕ can be expressed as the Taylor series expansion based on a specifically chosen boundary point (body intercept point $((x', y', z')|_{BI} = (0,0,0))$, with the following form:

$$\phi(x', y', z') \cong \phi_{BI} + \left.\frac{\partial \phi}{\partial x}\right|_{BI} x' + \left.\frac{\partial \phi}{\partial y}\right|_{BI} y' + \left.\frac{\partial \phi}{\partial z}\right|_{BI} z' + \frac{1}{2}\left.\frac{\partial^2 \phi}{\partial x^2}\right|_{BI} (x')^2 + \frac{1}{2}\left.\frac{\partial^2 \phi}{\partial y^2}\right|_{BI} (y')^2 + \cdots, \tag{10}$$

where $x' = x - x_{BI}, y' = y - y_{BI}, z' = z - z_{BI}$.

In practice, Eq. (10) is approximated by an Nth-order polynomial:

$$\phi(x', y', z') \approx \Phi(x', y', z') = \sum_{i=0}^{N}\sum_{j=0}^{N}\sum_{k=0}^{N} c_{ijk}(x')^i (y')^j (z')^k \quad i + j + k \leqslant N. \tag{11}$$