Powder Technology 281 (2015) 99–111

Contents lists available at ScienceDirect

Powder Technology

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

Particle scale studies of heat transfer in a moving bed

W.J. Yang [a,b], Z.Y. Zhou [a,c,*], A.B. Yu [c]

[a] Laboratory for Simulation and Modelling of Particulate Systems, School of Materials Science and Engineering, The University of New South Wales, Sydney, NSW 2052, Australia
[b] Science and Technology on Combustion, Internal Flow and Thermal-Structure Laboratory Northwestern Polytechnical University, Xi'an 710072, PR China
[c] Laboratory for Simulation and Modelling of Particulate Systems, Department of Chemical Engineering, Monash University, Clayton, VIC 3800, Australia

ARTICLE INFO

Article history:
Received 10 December 2014
Received in revised form 31 March 2015
Accepted 26 April 2015
Available online 6 May 2015

Keywords:
Discrete element method
Heat transfer
Moving bed
Gas–solid flow

ABSTRACT

Moving beds acting as bed reactors are quite common in industry. Understanding of the multiphase flow and thermal behaviour in moving beds is important for process design and optimisation. In this work, CFD–DEM approach is adopted to investigate the heat transfer behaviour in a moving bed which is relevant to the raceway region in an ironmaking blast furnace. The solid flow behaviour including flow pattern, velocity, and microscopic properties are investigated. A good agreement of gas temperature distribution between simulation and experiments is achieved. Then the heat transfer behaviour is analysed quantitatively through heat flux and contribution of each heat transfer mode. It reveals that under the conditions considered in this work, particle-fluid convection is dominant, and the contribution of radiation is very small but increases when the bed temperature is high. Solid flow rate and gas flow rate have significant but different influences on the momentum and thermal behaviour in moving beds. The study presented in this work provides a solid base for the further investigation of thermal behaviour in moving beds with the consideration of chemical reactions, and more complicated operational conditions.

1. Introduction

Moving beds are often counter-current devices in which solids move downward by gravity and gas steam flows upward through the bed voids [1]. Moving beds have been widely used as chemical reactors, and typical examples are coal gasification reactor and ironmaking blast furnace. In a coal gasification reactor, coke particles descend to the raceway for combustion with gas at the bottom. For a blast furnace, ore particles are reduced to liquid iron by counter-current reducing gas in a range of temperature 1200–1400 K, and the coke combustion occurs in the raceway which locates near the gas injection [2–4]. The multiphase flow and heat transfer significantly influence reactor performance. To achieve optimal design and control of such moving bed reactors, it is important to understand the mechanisms of gas–solid flow and heat transfer characteristics [5].

Over the past decades, many studies have been carried out on the heat transfer in moving beds by physical experiments [6–14]. The experimental study often suffers the limitations from experimental techniques and instrumentation, but some significant progress has been made on the heat transfer measurements [6–9]. For example, Akiyama et al. [6] adopted a laboratory-scale moving bed to determine the effective thermal conductivity through the measurement of the thermal diffusivity by the laser flash method. Schaefer [10] described the reaction by an ignition temperature and the heat generated on a volume basis, and the temperature distribution of solid and gas in a moving bed. But these studies are generally at the macroscopic level, and it is often difficult to quantify the heat transfer fundamentals. In recent years, heat transfer behaviour at a microscopic, individual particle level has been examined experimentally in fluidized beds [11–14]. Such particle scale studies are useful to validate numerical models, but still not enough for understanding of fundamentals, i.e. identifying the different heat fluxes through individual particles.

Mathematical modelling has increasingly become an effective tool in investigating multiphase flow and heat transfer in fluid bed reactors. Generally, two approaches are widely used: continuum-based approach (e.g. two fluid model — TFM), and discrete-based approach (e.g. combined DNS and DEM, or combined CFD and DEM). They have been reviewed by many investigators [15–17]. The continuum approach on the basis TFM is an important approach to study heat transfer in fluid bed reactors [18–22], particularly for process modelling and applied research due to its computational convenience. However, this approach is often limited by homogeneity assumptions, constitutive equations for solid stress tensor, and difficulty in generating microscopic information for fundamental understanding. The combined DNS–DEM is a powerful method to obtain detailed results of hydrodynamic interactions between fluid and particles [23]. Heat transfer between gas and particles has been studied by DNS method, as reviewed by Deen et al. [24]. However, DNS has extremely high request of computing cost and often is limited to small systems. Its capacity in handling particle collisions is

* Corresponding author at: Laboratory for Simulation and Modelling of Particulate Systems, Department of Chemical Engineering, Monash University, Clayton, VIC 3800, Australia. Tel.: +61 3 9905 0846.
E-mail address: zongyan.zhou@monash.edu (Z.Y. Zhou).

http://dx.doi.org/10.1016/j.powtec.2015.04.071
0032-5910/

another major weakness of this model. As pointed out by Yu and Xu [25], at this stage of development, the difficulty in particle–fluid flow modelling is mainly related to solid phase rather than fluid phase. Therefore, because of its superior computational convenience as compared to DNS and capability to capture the particle physics as compared to TFM, the combined CFD–DEM approach becomes more attractive [26–29]. By CFD–DEM, information such as particle–particle or particle–wall contact, local voidage and local gas–solid flow structure can be produced. Such information is essential in determining the heat transfer behaviour of individual particles. The approach has been used by many investigators to study coal combustion [30,31], air drying [32], olefin polymerization [33] and the heat transfer in packed and fluidized bed [34–36]. A number of attempts have also been made to study multiphase flow in a blast furnace by CFD–DEM [37–41], but few studies are carried out on the heat transfer.

In this work, the CFD–DEM approach implemented with heat transfer is proposed to investigate the thermal behaviour in moving beds which is relevant to the ironmaking blast furnace raceway region. Hot gas is injected laterally from one side and particles are discharged from the bottom near the gas inlet. The flow and heat transfer patterns are simulated, and compared with literature data. Different heat transfer mechanisms are quantitatively evaluated to capture the main features of the heat transfer in the moving bed, which is important and meaningful in the understanding of some industrial process, i.e. blast furnace. Then the effects of gas and solid flow rate are studied and discussed in detail.

2. Model description

DEM used in this work is based on the so called soft sphere approach [42]. It is coupled with CFD, and heat transfer is implemented into CFD–DEM. A full model description can be found in the literature [34]. For convenience, it is given below.

2.1. Governing equations for particles

The governing equations for the translational and rotational motions, and the energy conservation of particle i with radius R_i, mass m_i and moment of inertia I_i can be written as:

$$m_i \frac{d\mathbf{v}_i}{dt} = \mathbf{f}_{pf,i} + \sum_{j=1}^{k_c} (\mathbf{f}_{c,ij} + \mathbf{f}_{d,ij}) + m_i \mathbf{g}. \tag{1}$$

$$I_i \frac{d\boldsymbol{\omega}_i}{dt} = \sum\nolimits_{j=1}^{k_c} (\mathbf{M}_{t,ij} + \mathbf{M}_{r,ij}). \tag{2}$$

$$m_i c_{p,i} \frac{dT_i}{dt} = \sum_{j=1}^{k_i} Q_{i,j} + Q_{i,f} + Q_{i,rad} \tag{3}$$

where $\mathbf{v}_i$ and ω_i are the translational and angular velocities of the particle, respectively, and k_c is the number of particles in interaction with the particle i. The forces involved are: the gravitational force $m_i\mathbf{g}$, and inter-particle forces which include elastic force $\mathbf{f}_{c,ij}$, viscous damping force $\mathbf{f}_{d,ij}$, and particle–fluid interaction force $\mathbf{f}_{pf,i}$. The torques acting on particle i by particle j include: $\mathbf{M}_{t,ij}$ generated by tangential force, $\mathbf{M}_{r,ij}$ commonly known as the rolling friction torque. Their expressions have been listed in the previous publications [26,34,39–41]. In the energy governing equation, k_i is the number of particles exchanging heat with particle i, $Q_{i,j}$ is the heat flux between particles i and j due to conduction, $Q_{i,f}$ the heat flux by convection between particle i and its local surrounding fluid, and $Q_{i,rad}$ the heat flux between particle i and its surrounding environment by radiation.

2.2. Governing equations for gas flow

The continuum fluid field is calculated from the continuity and Navier–Stokes equations based on the local mean variables over a computational cell, which can be written as:

$$\frac{\partial \varepsilon_f}{\partial t} + \nabla \cdot (\varepsilon_f \mathbf{u}) = 0. \tag{4}$$

$$\frac{\partial \left(\rho_f \varepsilon_f \mathbf{u}\right)}{\partial t} + \nabla \cdot \left(\rho_f \varepsilon_f \mathbf{u}\mathbf{u}\right) = -\nabla p - \frac{\sum_{i=1}^{k_V} \mathbf{f}_{pf,i}}{\Delta V} + \nabla \cdot \varepsilon_f \boldsymbol{\tau} + \rho_f \varepsilon_f \mathbf{g}. \tag{5}$$

$$\frac{\partial \left(\rho_f \varepsilon_f c_p T\right)}{\partial t} + \nabla \cdot \left(\rho_f \varepsilon_f \mathbf{u} c_p T\right) = \nabla \cdot (c_p \Gamma \nabla T) + \frac{1}{\Delta V}\left(\sum_{i=1}^{k_V} Q_{f,i} + Q_{f,wall}\right) \tag{6}$$

where $\mathbf{u}$, ρ_f, and p are the fluid velocity, density and pressure, respectively, and k_V is the number of particles in a computational cell of volume ΔV. $\boldsymbol{\tau}$ and ε_f are the fluid viscous stress tensor and porosity, respectively. $Q_{f,i}$ is the heat flux between fluid and particle i which locates in a computational cell, and $Q_{f,wall}$ is the fluid–wall heat flux.

2.3. Heat transfer models

As shown in Eqs. (3) and (6), three heat transfer mechanisms are considered in the present CFD–DEM model: fluid–particle or fluid–wall convection; particle–particle or particle–wall conduction; and particle radiation. The models used to calculate different heat fluxes have been well described somewhere [28,34–36]. For convenience, they are briefly given below.

2.3.1. Convective heat transfer

The convective heat transfer rate between particle i and fluid is calculated according to $Q_{i,f} = h_{i,conv} \cdot A_i \cdot (T_{f,i} - T_i)$, where A_i is the particle surface area, $T_{f,i}$ is fluid temperature in a computational cell where particle i is located, and $h_{i,conv}$ is the convective heat transfer coefficient. $h_{i,conv}$ is associated with the Nusselt number, which is a function of particle Reynolds number, Re_i, and gas Prandtl number, Pr, given by

$$Nu_i = h_{i,conv} d_{pi}/k_f = 2.0 + a\mathrm{Re}_i^b \,\mathrm{Pr}^{1/3}. \tag{7}$$

Pr is assumed to be a constant in this work, set to 0.712 corresponding to that for air at 300 K. a and b are two parameters that need to be evaluated, in the present simulation, $a = 1.2$, $b = 0.5$, is adopted. The particle Reynolds number can be calculated by $\mathrm{Re}_i = \rho_f d_{pi} \varepsilon_i |\mathbf{u}_{gi} - \mathbf{v}_{pi}|/\mu_f$, where ρ_f and μ_f are gas density and molecular dynamic viscosity, respectively; d_{pi} and ε_i are the diameter of particle i and the porosity around particles i; $\mathbf{u}_{gi}$ and $\mathbf{v}_{pi}$ respectively are the gas velocity around particle i and the velocity of particle i. The Reynolds number in the present work varies from 1000 to 7000. For fluid–wall heat transfer, $Nu_D = h_{f,wall} D/k_f = 0.023\mathrm{Re}^{0.8}\,\mathrm{Pr}^n$ is used to determine the heat transfer coefficient $h_{f,wall}$, where D is the hydraulic diameter, and the exponent n is 0.4 for heating, and 0.3 for cooling. Then the heat flux $Q_{f,wall}$ between particle and wall is determined by $Q_{f,wall} = h_{f,wall} \cdot A_{f,wall} \cdot (T_{wall} - T_f)$, where $A_{f,wall}$ is the contact area between fluid and wall. For the further details, see Ref. [34].

2.3.2. Conductive heat transfer

Conduction between particles mainly includes (i) particle–fluid–particle conductive heat transfer; and (ii) particle–particle conductive heat transfer, as indicated in Fig. 1. The model that details for the two mechanisms are described as follows.