Powder Technology 284 (2015) 475–485

Contents lists available at ScienceDirect

Powder Technology

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

Computational study of particle temperature in a bubbling spout fluidized bed with hot gas injection

A.V. Patil, E.A.J.F. Peters *, J.A.M. Kuipers

Multiphase Reactors Group, Department of Chemical Engineering & Chemistry, Eindhoven University of Technology, P.O. Box 513, 5600 MB Eindhoven, the Netherlands

ARTICLE INFO

Article history:
Received 13 February 2015
Received in revised form 26 June 2015
Accepted 14 July 2015
Available online 22 July 2015

Keywords:
Spout fluidized bed
Temperature distribution
Heat transfer

ABSTRACT

CFD–DEM simulations of a hot gas jet issuing into a pseudo 2D gas–solid fluidized bed of glass particles are reported. A novel CFD–DEM model accounting for fluid–particle heat transfer has been developed and used for this particular research. In this work the background gas is maintained at minimum fluidization velocity and room temperature (300 K) while a hot gas jet (500 K) is injected from the center of a pseudo 2D bed. Three different particle sizes have been studied (1 mm, 2 mm and 3 mm). The hot gas jet causes continuous bubble formation and propagation in the bed thus creating a circulation pattern of particles. Using the detailed simulation data various kinds of analysis are presented on the particle temperature statistics like standard deviation and distribution profiles of time-averaged particle density, volume flux and temperature variation about the mean. Further, a tracer particle analysis is presented that shows the variation of particle temperatures about the mean for different particle sizes.

1. Introduction

Spouted fluidized beds are widely used in processes such as granulation, drying and coating. Some of the products produced by such processes are detergents, pharmaceuticals, food and fertilizers. These products need to be produced with specific properties such as size, mechanical strength (for ease of handling) and chemical composition (purity). Fluidization using spouted beds is widely considered to be the most suited mode of operation to produce granular products. This has resulted in a large amount of research on spout fluidized bed applications in literature [1–4].

The spout, which has a high speed gas jet compared to the background fluidization gas, avoids creation of slugs and thus facilitates a well defined particle circulation pattern. This is believed to facilitate optimal and uniform particulate processing making this processing method one of the most popular and promising methods. A large number of experimental investigations have been carried out on spout fluidization to understand overall particle flow dynamics and their regimes of operations in spout fluidization; see [5–10]. Besides this, the spectral analysis of pressure drop fluctuations has been exhaustively studied in literature [11–13]. Hydrodynamics of spouted fluidized beds has been studied extensively both experimentally and computationally in the past using relatively simple one dimensional models [14–16].

Early modeling works on spout fluidizations involved many kinds of theoretical and population balance models for size distributions [17,18]. Individual particle flow modeling first became possible with the advent of the discrete particle model (DEM) by Tsuji et al. [19]. The application of discrete particle models to study spout fluidized beds was introduced by Kawaguchi et al. [20]. This further led to many studies using improved gas–particle and particle–particle modeling methods for spout fluidized beds which is now known as the computational fluid dynamics–discrete particle model (CFD–DEM) technique [21–23].

CFD–DEM is a Euler–Lagrange based modeling tool which is best suited for medium scale studies of gas–solid fluidized bed systems; see Link [24]. Thus it is often used to assess the impact of closures for fluid–particle and particle–particle interaction models in various spouting regimes backed up by novel experimental techniques [25,26]. Some recent works with CFD–DEM have focussed on improving particle–particle interaction models that produce results that better correspond to experimental observations [27,28]. Experimental validations of spouted fluidized beds have also been extended to multiple and elevated spout studies [29].

In CFD–DEM the gas phase is treated as a continuum (Eulerian) whereas the particulate phase as discrete (Lagrangian) fashion. Through two-way-coupling both phases experience each other through momentum exchange. Particle collision mechanics is accounted for using the soft sphere model first proposed by Cundall and Strack [30]. CFD–DEM models have been extended with heat transfer descriptions where the continuous phase is modeled by a convection–diffusion equation whereas the discrete phase is treated using a thermal energy equation for each particle.

Heat transfer to the emulsion phase in spouted fluidized beds is of great importance as it determines the efficiency of various process operations like drying, coating, etc. With the extension of energy transfer in DEM heat transfer in spout fluidized bed systems [31,32] can be

* Corresponding author. Tel.: +31 40 247 3122; fax: +31 40 247 5833.
E-mail address: e.a.j.f.peters@tue.nl (E.A.J.F. Peters).

http://dx.doi.org/10.1016/j.powtec.2015.07.014
0032-5910/

studied in detail. Development of new infrared techniques has lead to possibilities of experimental validations of these models using detailed temperature distributions [33–35].

Studies of spout fluidized beds are not limited to thermal aspects but can be extended to include the motion (mixing) of individual particles. The motion of tracer particles in fluidized beds has been researched extensively in literature using PEPT measurements [36–38]. However these studies are limited to individual particle motion measurement. So for studying particle temperature variations during these motions or circulations the CFD–DEM approach can be used. CFD–DEM with heat transfer brings many possibilities of studies in spout fluidized beds such as particle size effects on both hydrodynamic and thermal behaviors.

In this work we study heat transfer to the particle emulsion from a hot gas jet that is injected at the center bottom of a pseudo 2D fluidized bed. A typical spouting process has various regimes of operations and here we study the continuous bubbling regime. In this regime continuous formation of hot gas bubbles takes place at the nozzle. Once the bubble reaches a certain size it detaches and rises through the bed. Depending on the particle size and background gas velocity the injected bubbles exchange heat with the emulsion phase at different rates.

This paper aims to study variations in particle distribution profiles for spout fluidization in bubbling regime operation as a function of particle size. Properties of particles like overall particle temperature standard deviation and spatial distribution have been studied. These properties in general were observed to increase with increasing particle size.

Tracer particle study is a novel method that has been previously employed in experiments and simulations [39,40]. Mostly these studies have been limited to tracer particle position study and did not involve properties like temperature of particles. Here a tracer particle study was performed to study individual particle behavior with respect to the mean particle temperature.

We will start with stating the governing equations in Section 2 and provide details of the spout injection system used for our study in Section 3. In Section 4 the main results will be presented, the main conclusions will be summarized in Section 5.

2. Governing equations

2.1. Gas phase modeling

In our CFD–DEM model the gas phase is treated as a continuum. The volume-averaged conservation equations for mass and momentum are respectively given by,

$$\frac{\partial}{\partial t}\left(\varepsilon_f \rho_f\right) + \nabla\cdot\left(\varepsilon_f \rho_f \mathbf{u}\right) = 0, \tag{1}$$

$$\frac{\partial}{\partial t}\left(\varepsilon_f \rho_f \mathbf{u}\right) + \nabla\cdot\left(\varepsilon_f \rho_f \mathbf{u}\mathbf{u}\right) = -\varepsilon_f \nabla p - \nabla\cdot\left(\varepsilon_f \tau_f\right) + \mathbf{S}_p + \varepsilon_f \rho_f \mathbf{g}, \tag{2}$$

where $\mathbf{S}_p$ denotes the source term for momentum transfer from the particulate phase and is given by

$$S_p = \sum_a \frac{\beta V_a}{1-\varepsilon_f}(\mathbf{v}_a - \mathbf{u})\delta(\mathbf{r}-\mathbf{r}_a) \equiv a - \beta\mathbf{u}. \tag{3}$$

Here β represents the volumetric drag coefficient for particles due to gas flow whereas α is the momentum felt by the gas phase per unit volume due to the overall particle motion. The particle properties are distributed on the Eulerian grid using smoothed Dirac-delta functions. The drag coefficient, β, is obtained from the Ergun [41] equation for the dense regime and the Wen and Yu [42] equation for the dilute regime;

$$F_{\text{drag}} = \frac{\beta\, d_p^2}{\mu} = \begin{cases} 150\,\dfrac{1-\varepsilon_f}{\varepsilon_f} + 1.75\,(1-\varepsilon_f)\,\mathrm{Re}_p & \text{if } \varepsilon_f < 0.8 \\ \dfrac{3}{4} C_D\, \mathrm{Re}_p\,(1-\varepsilon_f)\,\varepsilon_f^{-2.65} & \text{if } \varepsilon_f > 0.8 \end{cases}. \tag{4}$$

The gas phase flow solver uses the semi-implicit projection method. The details of this method can be found in Patil et al. [32] and hence will not be reported here.

The thermal energy equation for the gas phase is given by,

$$\frac{\partial\left(\varepsilon_f \rho_f C_{p,f} T\right)}{\partial t} + \nabla\cdot\left(\varepsilon_f \rho_f \mathbf{u} C_{p,f} T\right) = \nabla\cdot\left(\varepsilon_f k_f^{\text{eff}} \nabla T\right) + Q_p, \tag{5}$$

where Q_p represents the source term from heat coupling with the particulate phase and k_f^{eff} is the effective thermal conductivity of the gas phase which is related to the intrinsic fluid thermal conductivity given by the following equation,

$$k_f^{\text{eff}} = \frac{1-\sqrt{1-\varepsilon_f}}{\varepsilon_f} k_f. \tag{6}$$

This equation was first proposed by Syamlal and Gidaspow [43]. The fluid–particle heat transfer can be obtained by summing the contributions of all particles belonging to a particular Eulerian cell using a smoothed delta-function as,

$$Q_p = -\sum_a Q_a \delta(\mathbf{r}-\mathbf{r}_a) = \sum_a h_{fp} A_a (T_a - T_f)\delta(\mathbf{r}-\mathbf{r}_a) \tag{7}$$

Where Q_a is the heat transferred from the gas to particle a and h_{fp} is the gas–particle heat transfer coefficient for which we use the empirical correlation given by Gunn [44],

$$\begin{aligned} \mathrm{Nu}_p = &\left(7 - 10\,\varepsilon_f + 5\,\varepsilon_f^2\right)\left[1 + 0.7\,\mathrm{Re}_p^{0.2}\,\mathrm{Pr}^{0.33}\right] \\ &+ \left(1.33 - 2.40\,\varepsilon_f + 1.20\,\varepsilon_f^2\right)\mathrm{Re}^{0.7}\,\mathrm{Pr}^{0.33} \end{aligned} \tag{8}$$

where,

$$\mathrm{Nu}_p = \frac{h_{fp}\, d_p}{k_f}, \mathrm{Re}_p = \frac{d_p\, \varepsilon_f\, \rho_f\, |\mathbf{u}-\mathbf{v}|}{\mu_f} \quad \text{and} \quad \mathrm{Pr} = \frac{\mu_f\, C_{p,f}}{k_f}. \tag{9}$$

2.2. Discrete particle phase

The modeling of the particle phase flow is based on the Cundall and Strack [30] model of the resultant force acting on individual spherical particles due to various contributing factors. Any given particle a with mass m_a and moment of inertia I_a is described by Newton's equations:

$$m_a \frac{\mathrm{d}^2 \mathbf{r}_a}{\mathrm{d}t^2} = -V_a \nabla p + \frac{\beta V_a}{1-\varepsilon_f}(\mathbf{u}-\mathbf{v}_a) + m_a\,\mathbf{g} + F_{\text{contact},a} \tag{10}$$

$$I_a \frac{\mathrm{d}\omega_a}{\mathrm{d}t} = \tau_a, \tag{11}$$

where $\mathbf{r}_a$ is the position. The forces on the right-hand side of Eq. (10) are the pressure gradient, drag, gravity and contact forces due to collisions, τ_a is the torque, and ω_a the angular velocity vector. The major contributions to the torque on a particle are the tangential forces experienced while colliding with other particles.

The heat transfer rate from the fluid to the particles is evaluated as the product of the fluid–particle heat transfer coefficient and the driving