Chemical Engineering Journal 150 (2009) 374–384

Contents lists available at ScienceDirect

Chemical Engineering Journal

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

Numerical study on the cluster flow behavior in the riser of circulating fluidized beds

Huanpeng Liu*, Huilin Lu

School of Energy Science and Engineering, Harbin Institute of Technology, Harbin 150001, China

ARTICLE INFO

Article history:
Received 30 October 2008
Received in revised form 3 January 2009
Accepted 8 January 2009

Keywords:
CFBs
Cluster
DSMC method
LES
Low-pressure zone

ABSTRACT

In this paper, the cluster flow behavior in the riser of circulating fluidized beds (CFBs) was predicted by means of the Euler–Lagrange approach. Gas turbulence was modeled by means of Large Eddy Simulation (LES). Particle collision was modeled by means of the direct simulation Monte Carlo (DSMC) method. An extended cluster identification method was used to obtain the solid concentration and velocities of clusters. The flow behavior of falling clusters in the near wall region and the up-flowing and down-flowing clusters in the core of riser were analyzed respectively. At the same time, some cluster transient flow characteristics observed in the CFBs experimental studies were obtained in present simulation. Simulated results showed that there exists a low-pressure zone between the tails of a down-flowing cluster in the core of riser and a new cluster can be formed in the near wall region and transferred from the near wall region to the core of riser under the effect of upgoing gas. Simulated results have a reasonable agreement with the previous experimental findings.

1. Introduction

One important feature of hydrodynamic behavior in circulation fluidized beds (CFBs) is the existence of clusters. A cluster is a group of particles held together as a result of hydrodynamic. They will affect the macro gas–solid flow behaviors in the reactor [1]. The study of clusters has received a great deal of attention during the last decades. For example, Van Den Moortel et al. [2] investigated the hydrodynamic characteristics of solid phase in the riser of CFBs using a phase Doppler particle analyzer (PDPA). Sharma et al. [3] investigated the effect of particle size and superficial velocity on the duration time, occurrence frequency, time-fraction of existence and solid concentration of clusters using capacitance-probe measurements in a fast-fluidized bed. Manyele et al. [4] study the cluster behavior in a high-density and high-flux CFB riser. At the same time, significant research efforts have been made to develop detailed physical models to study the complex hydrodynamics of CFBs. Many investigators have used these numerical models to study the flow behavior of cluster in the riser of CFBs and obtained some valuable findings. Broadly speaking, the simulation approaches of two-phase flow in circulating fluidized beds can be classified into Euler–Euler two-fluid model and Euler–Lagrange discrete particle trajectory model. In two-fluid model, gas and solid phases are both considered as continuous mediums, and balance equations of each phase are established to investigate the flow behavior of gas and solid phases. Tsuo and Gidaspow [5] successfully simulated the flow pattern of clusters in the CFBs by means of two-fluid model. Lu et al. [6] predicted the flow behavior of clusters in CFBs by means of a proposed cluster-based approach (CBA). In the Euler–Lagrange particle trajectory model, gas is considered as the continuous medium and the motions of particles are treated in the Lagrange coordinate by solving the motion equations. Tsuji et al. [7] verified the ability of discrete particle modeling on the simulation of the formation of clusters in CFBs and found that there exist clusters in the core region of riser of CFBs. Ouyang and Li [8] studied the effects of gas velocity, solid flow rate, and inelastic collisions on cluster formation using Euler–Lagrange simulations. Helland et al. [9,10] studied the cluster structures and the fluctuating gas–solid motion in the CFBs. Yonemura and Tanaka [11] investigated the formation of clusters in the circulating fluidized bed by means of DSMC method. Effects of physical properties of particles on the structure of particle clusters were studied numerically in a rectangular domain with periodic boundaries [12]. A numerical simulation was performed for a dispersed gas–solid flow in a vertical channel by DSMC method [13]. It is found that the flow becomes unstable and inhomogeneous as the gas velocity decreases and the solid loading increase. Although a great deal of research on particle cluster characteristics has been performed in the last decades, some flow properties of clusters are still not understood very well.

In this study the locally averaged Navier–Stokes equations of gas phase and Lagrangian type particle motion equations are simul-

* Corresponding author.
E-mail address: liuhuanpeng@hit.edu.cn (H. Liu).

1385-8947/$ – see front matter © 2009 Elsevier B.V. All rights reserved.
doi:10.1016/j.cej.2009.01.015

Nomenclature

C_d	drag coefficient
d	particle diameter
e	restitution coefficient of particles
e_w	restitution of wall
g	gravity
g_0	redial distribution function
H	bed height
n	normal direction
N	the number of cluster occurrence
P	pressure
Re	Reynolds number
r	radial direction
t	tangential direction
μ_g	gas velocity
v_s	particle velocity
v_{cl}	cluster velocity
W	bed width

Greek letters

τ_g	gas stress tensor
τ_s	particle stress tensor
μ_g	gas viscosity
ε_g	void fraction
ε_{cl}	solid concentration of cluster
ρ_g	gas density
ρ_s	particle density

taneously solved. The mutual interaction between gas phase and particles is taken into account. Particle collision is modeled by means of the DSMC method. The variations of the mean velocity and solid concentration of clusters near the wall region and in the core of riser along the height of riser are analyzed respectively. The shape, occurrence frequency and existence time fraction of up-flowing and down-flowing clusters in the core of riser are compared. Some micro-scale transient flow characteristics of clusters are investigated in this study.

2. Eulerian–Lagrangian gas–solid flow model

2.1. Continuity and momentum equations for gas phase

The Euler–Lagrangian method computes the Navier–Stokes equation for the gas phase and the motion of individual particles by the Newtonian equations of motion. For the gas phase, we write the equations of conservation of mass and momentum [14]:

$$\frac{\partial}{\partial t}(\rho_g \varepsilon_g) + \nabla \cdot (\rho_g \varepsilon_g u_g) = 0 \quad (1)$$

$$\frac{\partial}{\partial t}(\varepsilon_g \rho_g u_g) + \nabla \cdot (\varepsilon_g \rho_g u_g u_g) = -\varepsilon_g \nabla P - S_{p-g} - \varepsilon_g (\nabla \cdot \tau_g) + \varepsilon_g \rho_g g \quad (2)$$

$$\tau_g = \mu_g [\nabla u_g + \nabla u_g^T] - \frac{2}{3}\mu_g \nabla \cdot u_g I \quad (3)$$

$$\mu_g = \mu_{lam,g} + \mu_t \quad (4)$$

where u_g and ρ_g are gas velocity and density, respectively. ε_g is the void fraction. The interaction forces between the two phases should be equal and have reserve directions. The value can be determined

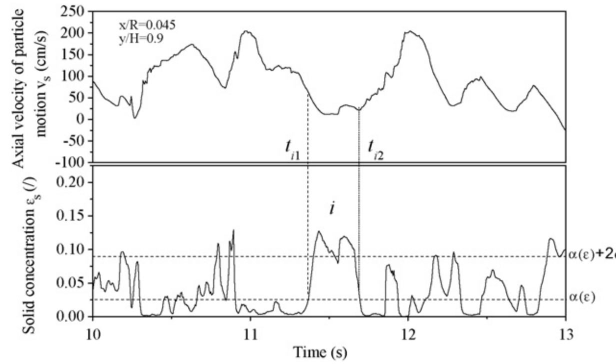

Fig. 1. Cluster identification method.

by:

$$S_{p-g} = \frac{\sum_{i=1}^{N} f_{d,i}}{S} \quad (5)$$

where $f_{d,i}$ is the interaction drag force acting on a particle.

Yuu et al. [15] has modeled turbulent viscosity coefficient of subgrid-scale turbulence caused by subgrid-scale fluctuations using LES in which the effect of particle oscillations on gas turbulence has been taken into account. The turbulent viscosity of gases is as follows:

$$\mu_t = C_V \Delta \rho_g k_g^{1/2} \quad (6)$$

where $\Delta = (xyz)^{1/2}$,and $k_g = (1/2)(\bar{u}'_{gx}\bar{u}'_{gy})$ is gas turbulent energy.

2.2. Particle motion equation

The particle motion is subjected to Newton's equation of motion. Magnus force, Saffman force, Basset, and the unsteady force are neglected due to the high ratio of particle density to gas density. The equation of translational motion of a particle can be written as follows [16]:

$$m\frac{dv_i}{dt} = -\frac{\pi}{6} d_i^3 \nabla P + f_d + m_i g \quad (7)$$

$$f_d = \left(\frac{1}{16}\right) C_{d0,i} \rho_g \pi d_i^2 |u_{gi} - v_i| (u_{gi} - v_i) \varepsilon_g^{-\delta} \quad (8)$$

where the drag force coefficient $C_{d0,i}$ is written as

$$C_{d0,i} = \left(0.63 + \frac{4.8}{Re_{p,i}^{0.5}}\right)^2 \quad (9)$$

$$Re_{p,i} = \rho_g d_i \frac{|u_{gi} - v_i|}{\mu_g} \quad (10)$$

$$\delta = 3.7 - 0.65 \ \exp\left[-\frac{(1.5 - \log_{10} Re_{p,i})^2}{2}\right] \quad (11)$$

The equation of rotational motion of a particle is written as

$$\frac{m_i d_i^2}{10}\frac{d\omega_i}{dt} = \frac{\rho_s d_i^2}{64}\left(\frac{6.45}{Re_\omega} + \frac{32.1}{Re_\omega}\right)|\omega_i|\omega_i \quad (12)$$

where $Re_\omega = d_i^2 \rho_g |\omega| / (4\mu_g)$.

2.3. Particle collision dynamics

The changes of velocity after a collision between two particles are subject to the equations [9,15,16]:

$$m_i(v_{i,1} - v_{i,0}) = J \quad (13)$$