Computers and Chemical Engineering 77 (2015) 135–146

Contents lists available at ScienceDirect

Computers and Chemical Engineering

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

Coupling of smoothed particle hydrodynamics and finite volume method for two-dimensional spouted beds

Fuzhen Chen*, Hongfu Qiang, Weiran Gao

601 Staff Room, Xi'an Hi-Tech Institute, Xi'an 710025, China

ARTICLE INFO

Article history:
Received 26 May 2014
Received in revised form 10 March 2015
Accepted 5 April 2015
Available online 11 April 2015

Keywords:
Smoothed particle hydrodynamics
Finite volume method
Coupled method
Spouted bed
Particle dynamics
Fluidization

ABSTRACT

A coupled method with smoothed particle hydrodynamics (SPH) and finite volume method (FVM) is proposed in this work for the simulation of the particle dynamics in two-dimensional spouted beds. Based on the pseudo-fluid model, SPH is used for discrete phase to trace the movement of each individual particle and FVM for continue phase to compute the turbulent fluid. Two phases are coupled through effects of drag force, gas pressure and volume fraction of each phase. A two-dimensional tapered-based spouted bed is chosen as a case study to demonstrate the performance of the SPH–FVM coupled algorithm. The simulation results show a good agreement with the experimental data and other simulation results by the two-fluid model and discrete element method in the literature. The spouted shape, time-averaged particle velocities and particle vertical velocities in the spout are analyzed and the distribution of gas flow field and turbulent kinetic energy are then discussed. It indicates that the present method is more suitable to study the fluidization within the spouted beds.

1. Introduction

Spouted beds, providing good mixing and contacting of particles and gas flows, have been widely used in many industry fields, such as agriculture, petrifaction, energy, foodstuff, medicine, metallurgy and so on. Spouted beds were originally invented in Canada by Mathur and Gishler (1995), the shape of which is usually conical based. In recent years, many researchers have recognized the distinctive advantages of spouted beds as reactors for various chemical processes. For example, the toroidal nature of particle motion in spouted beds makes it easy to recirculate heat in an organized manner, thus leading to extended ranges of flammability, enhanced reaction rates, and super-adiabatic temperatures, considerably higher than the adiabatic flame limit of premixed reactants for a given preheat temperature. Therefore, spouted beds have been gradually applied as chemical reactors, including as combustion reactors, coal gasification reactors, catalytic partial oxidation reactors, catalytic oxidative coupling reactors, catalytic polymerization reactors, and pyrolysis reactors. The granular phase in spouted beds is agitated by the gases through a single nozzle. Spouted beds are usually divided into three different regions: a region of upward gas–solid flow called the spout, another region of downward quasi-static granular flow called the annulus and the other region called the fountain (Mathur and Epstein, 1974). Particles move in a cycle in these three regions. Since the better characters in gas–solid contacting efficiency and fast agitation than traditional fluidized beds, it is necessary to study the particle motions and gas flow patterns of the spouted beds.

Experiments and simulations are the main methods to study the spouted beds. Optical fiber systems have been extensively used by many researchers to determinate the trajectories, velocity and recirculation time of particles in spouted beds (Randelman et al., 1974; Benkrid and Caram, 1989; He et al., 1994; San José et al., 1998, 2005; Zhao et al., 2006). He et al. (1994) used a fiber optic probe system to measure the vertical particle velocity profiles in the spout, annular and fountain regions of a full-column spouted bed. Benkrid and Caram (1989) also used this technique to measure particle velocities in the annulus of a full-column spouted bed. They found there was a plug flow zone in the upper part of the annulus. Zhao et al. (2006) measured the vertical particle velocity profiles in a full-column cylindrical conical spouted bed with and without a draft tube with this experimental system. The differences between the two kinds of spouted beds were investigated. This technique has been used widely, but it can bring disturbance to the local flow field when introducing optical fibers. Though it allows for non-invasively measurement of particle velocities through the flat walls for semi-column spouted beds, they were found not to apply to full-column spouted beds (Duarte et al., 2009). Radioactive particle tracking (RPT), as a non-intrusive technique, is based on the principle of detecting positions of tracer particles continuously (Roy

---

* Corresponding author. Tel.: +86 29 15829728486.
E-mail address: chen_fu_zhen@163.com (F. Chen).

http://dx.doi.org/10.1016/j.compchemeng.2015.04.002
0098-1354/

et al., 1994; Djeridane et al., 1998; Larachi et al., 2003). Roy et al. (1994) used a γ-ray-emitting particle tracking technique to measure the particle velocities in a spouted bed. Djeridane et al. (1998) used RPT technique to investigate the mean and turbulent particle velocity fields in a spouted bed. Larachi et al. (2003) also studied the characters of mixing and circulation of solids in the spouted bed with this technique. For its negligible disturbance to the solids and gas flows, it can be used to improve the understanding of particle turbulent motions and their modeling in spouted beds. However, RPT is unsuitable for the precise measurement of dynamic or periodic behavior of solid particles for its limitations in tracing particles at one time. Particle image velocimetry (PIV), different from both the optical fiber system and RPT techniques, can measure instantaneous velocity fields non-intrusively within global flow domain (Liu et al., 2008; Zhao et al., 2008). Liu et al. (2008) used the PIV technique with a self developed algorithm to investigate particle flow behaviors in a 2D spouted bed. They discussed the periodic particle flow pattern, the time-averaged particle velocity profiles in both the spout and the annulus and the granular temperature distribution of the entire bed. Zhao et al. (2008) investigated the particle velocity distributions from digital images of the whole bed flow field with the PIV method. These experimental results were used to validate their DEM simulation results. The simulation results in this paper are also verified with these results by Zhao et al. (2008).

Although the information of particles' velocities, trajectories and temperatures and so on can be attained through experiments, the details of motion of particles and gas flow field captured by experiments are limited. In addition the conditions of experiments are very strict such as some spouted beds used as chemical equipments should be operated at high temperature and pressure. Furthermore, it is costly to do an experiment. So the approaches based on computational fluid dynamics (CFD) have been widely used for studying gas–solid two-phase flow, with an advantage of easily obtaining detailed information and wide range flow properties. So far almost all simulations on gas–solid two-phase flow are based on two methods that are two-fluid model (TFM) and the discrete particle/element method (DPM/DEM). In the DPM/DEM method, the gas phase is described by a locally averaged Navier–Stokes equation, whereas the motion of each individual particle is traced by a soft-sphere or hard-sphere model, and two phases are coupled by interphase forces. This approach offers a nature way to simulate the system of spouted bed with complex behaviors. Many researchers (Takeuchi et al., 2005; Kawaguchi et al., 2000; Swasdisevi et al., 2004) have used the DEM approach to study the granular dynamics in spouted beds. The flow characters of particles in the bed are obtained. Krzywanski et al. (1992) developed a multi-dimensional model to describe the gas and particle dynamic behavior in a spouted bed. Kawaguchi et al. (2000) proposed an Euler–Lagrange approach, in which the three-dimensional motion of solids was discretely traced by solving the Newton's equation of motion using DEM approach. Zhao et al. (2008) presented a DEM simulation on the incoherent spouting of solids in a 2D spouted bed. The particle velocity profiles are validated by experiments and the particle force, particle concentrations and gas flow profiles are all discussed. The methods above for gas–solid two-phase flow belong to Euler–Lagrange coupled approach. The major drawback of this approach is it requires considerable computational effort and the volume fraction is limited to only 10% for DPM.

For the other method, TFM, the different phases are mathematically treated as interpenetrating continua, and the conservation equations for each of the two phases are derived to obtain a set of equations that have a similar structure for each phase. Lu et al. (2001) presented a two-fluid gas–solids flow model for spouted beds, viewing spout and annulus as two interconnected regions. Lu et al. (2004) and Wang et al. (2009) introduced a kinetic–frictional constitutive model for dense assemblies of solids in the simulation of spouted beds. The model treated the kinetic and frictional stresses of particles additively. Du et al. (2006a, 2006b) used the TFM method to describe the influences of drag coefficient correlations, frictional stress, maximum packing limit and coefficient of restitution of particles on the simulation of spouted beds. Duarte et al. (2009) used the Eulerian multiphase model to simulate two kinds of spouted beds: a cylindrical vessel with a tapered bottom and a conical spouted bed. The methods above for gas–solid two-phase flow belong to Euler–Euler coupled approach. Although this approach is more feasible for practical applications to complex multiphase flows, the discrete profiles of the solid phase such as the velocity and trajectory of an individual particle can not be got.

Base on pseudo-fluid model, this paper presented a new method, called coupling of smoothed particle hydrodynamics and finite volume method (SPH–FVM), to simulate the spouted beds. With our new approach, each individual particle can be traced and the information of each particle like the diameter, mass, velocity, temperature and trajectory of each particle, can be gained. Otherwise, it can also decrease the computational cost. Furthermore, the volume fraction of the particles can vary from 0% to 100%. Due to the feature of unlimited volume fraction, this new method can be used in many fields from dilute to dense gas–solid two-phase flow such as ultra-rapid gas-phase catalytic reactions, sand storm, powder snow avalanche, pneumatic transport of particulate, cyclone separators and classifiers, chemical reactors, and fluidized beds. These systems often involve complicated flow dynamics and interactions between flow constituents and their surroundings. In this paper, firstly we describe the coupled method base on the pseudo-fluid model. Then this method is used to simulate a 2D spouted bed reported in the literature (Zhao et al., 2008). The spouted shape, time-averaged particle velocity and particle vertical velocities in the spout and annulus are compared with experimental data and the distribution of gas flow field and turbulent kinetic energy are then discussed.

2. Numerical method

SPH is a Lagrangian particle method introduced by Lucy (1977) and Gingold and Monaghan (1977) in 1970s in order to solve hydrodynamic problems in astrophysics. For its advantages of self adaptability, meshless, Lagrangian and particle properties, it has been extensively applied in study of galaxy dynamics, fluid dynamics, impact dynamics and micro-fluid dynamics (Monaghan, 2005) and so on. FVM, as an Euler grid-based method, has high precision and efficiency on the fluid dynamics filed. Through importing pseudo-fluid model, the coupled method with SPH and FVM is established. The solid phase is discretized by SPH particles whereas the gas phase is solved with FVM. The two methods are connected through drag force, pressure gradient and volume fraction of each phase.

2.1. Conservation equations in continuous phase

The mass conservation equations for continuous phase g (i.e. gas) is

$$\frac{\partial}{\partial t}(\alpha_g \rho_g) + \nabla \cdot (\alpha_g \rho_g \vec{v}_g) = 0 \tag{1}$$

The momentum balance for the continuous phase is

$$\frac{\partial}{\partial t}(\alpha_g \rho_g \vec{v}_g) + \nabla \cdot (\alpha_g \rho_g \vec{v}_g \vec{v}_g) = -\alpha_g \nabla P + \nabla \cdot \bar{\bar{\tau}}_g + \vec{R}_{gs} + \alpha_g \rho_g \vec{g} \tag{2}$$

where α_g, ρ_g and $\vec{v}_g$ are the gas volume fraction, density and velocity, respectively, $\vec{R}_{gs}$ is an interaction force between phases. As mentioned in the literature (Zhao et al., 2008), the effects of