Particuology xxx (2014) xxx–xxx

Contents lists available at ScienceDirect

Particuology

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

Two and three dimensional modeling of fluidized bed with multiple jets in a DEM–CFD framework

Surya Deb, Danesh K. Tafti*

Randolph Hall (114 I), Department of Mechanical Engineering, Virginia Tech, Blacksburg, VA 24061, USA

ARTICLE INFO

Article history:
Received 17 December 2013
Received in revised form 2 April 2014
Accepted 2 April 2014

Keywords:
Discrete element modeling
Fluidized beds
Solid fraction
Bed height
Grid zone
Multiple jets
Particle flux

ABSTRACT

Fluidized beds with multiple jets have widespread industrial applications. The objective of this paper is to investigate the jet interactions and hydrodynamics of a fluidized bed with multiple jets. Discrete element modeling coupled with in-house CFD code GenIDLEST has been used to simulate a bed with nine jets. The results are compared with published experiments. Mono dispersed particles of size 550 μm are used with 1.4 times the minimum fluidization velocity of the particles. Both two and three dimensional computations have been performed. To the best of our knowledge, the results presented in this paper are the first full 3D simulations of a fluidized bed performed with multiple jets. Discrepancies between the experiment and simulations are discussed in the context of the dimensionality of the simulations. The 2D solid fraction profile compares well with the experiment close to the distributor plate. At higher heights, the 2D simulation over-predicts the solid fraction profiles near the walls. The 3D simulation on the other hand is better able to capture the solid fraction profile higher up in the bed compared to that near the distributor plate. Similarly, the normalized particle velocities and the particle fluxes compare well with the experiment closer to the distributor plate for the 2D simulation and the freeboard for the 3D simulation, respectively. A lower expanded bed height is predicted in the 2D simulation compared to the 3D simulation and the experiment. The results obtained from DEM computations show that a 2D simulation can be used to capture essential jetting trends near the distributor plate regions, whereas a full scale 3D simulation is needed to capture the bubbles near the freeboard regions. These serve as validations for the experiment and help us understand the complex jet interaction and solid circulation patterns in a multiple jet fluidized bed system.

1. Introduction

Jetting fluidized beds are widely used in various industries for good mixing and improved chemical reaction properties. The gas–solid contact in the region above the distributor plate of a jetting fluidized bed is improved by particle entrainment into the jets leading to improved mixing. Hence it is important to analyze the general flow hydrodynamics of the jetting zone. Several researchers have used experimental techniques to study the cold flow properties of jetting fluidized beds. A measurement of solid circulation in large jetting fluidized beds was conducted by Yang, Ettehadieh, and Haldipur (1986). In their study, they employed high speed motion capture along with a force probe to study the bubble characteristics. This was the first study attempted at measuring the bubble characteristics, simultaneously with solid circulation in a large commercial scale jetting fluidized bed. A systematic study of the motion of solids, solid flow rates and local void fraction fluctuations in a large jetting fluidized bed has been done by Ettehadieh, Yang, and Haldipur (1988). They analyzed local bed density fluctuations and determined that the solid circulation rate increases linearly with increasing jet velocities. The effects of a single bubble in the distributor region of a fluidized bed have been studied by Yates, Rowe, and Cheesman (1984). They used X-ray analysis to study the gas bubbles entering a fluidized bed reactor. The flow characteristics of a large jetting fluidized bed with two nozzles have been studied by Guo, Liu, and Zhang (2000). They proposed an empirical correlation for the jet penetration depth based on experimental data. Fluidization regime maps were developed by Sutanto, Epstein, and Grace (1985), including fixed, bubbling, jetting fluidized bed, spout fluidization, and spout with aeration. Flow regime

---

* Corresponding author. Tel.: +1 540 231 9975; fax: +1 540 231 9100.
E-mail address: dtafti@vt.edu (D.K. Tafti).

http://dx.doi.org/10.1016/j.partic.2014.04.005

Please cite this article in press as: Deb, S., & Tafti, D.K. Two and three dimensional modeling of fluidized bed with multiple jets in a DEM–CFD framework. *Particuology* (2014), http://dx.doi.org/10.1016/j.partic.2014.04.005

transitions and jet penetration depth were studied in a binary mixture setup by Guo, Yue, Zhang, and Liu (2001). They presented a new correlation for predicting the jet penetration depth for different annular gas flow rates. Significant amount of research has been conducted on a single jet (Basov, Markhevka, Melik-Akhnazarov, & Orochko, 1969; Huang & Chyang, 1991; Zhang, Zhang, Lovick, Zhang, & Zhang, 2002). Recently, Guo, Si, and Zhang (2010) investigated the jet penetration depth and particle concentration profiles using optical fiber probes and acoustic assistance.

Getting detailed information about the entire space of the bed is difficult in the experiments, which generally requires invasive procedures that can alter the flow and particle dynamics in the bed. A full scale numerical simulation is often desired to understand the complicated gas-particle dynamics inside a jetting fluidized bed. Recent advances in computational capabilities have given rise to a wide variety of modeling strategies to model granular flows. Among the available modeling approaches, the most widely used are the two fluid model (TFM) and the discrete element model (DEM). TFM is an Eulerian–Eulerian approach where both the fluid and the particles are considered to be two distinct interpenetrating continua. This approach is suitable for modeling large industrial scale gasifiers with low to moderate resolution of the flow field. DEM on the other hand is a high fidelity Eulerian–Lagrangian approach which tracks each individual particle, providing a much better flow field resolution compared to TFM. However, DEM is more suited for small lab scale simulations as it is restricted by the computational expense. DEM was first developed by Cundall and Strack (1979) and later coupled with CFD by Tsuji, Kawaguchi, and Tanaka (1993). Since then, DEM has found its place as a viable modeling approach in a wide variety of applications including segregation of binary mixtures (Feng, Xu, Zhang, Yu, & Zulli, 2004; Feng & Yu, 2010), capturing bubble dynamics (Kobayashi, Yamazaki, & Mori, 2000; Pain, Mansoorzadeh, Gomes, de Oliveira, & Goddard, 2002; Rong, Mikami, & Horio, 1999), modeling behavior of cohesive particles (Iwadate & Horio, 1998; Mikami, Kamiya, & Horio, 1998) and so on. A comprehensive review focused on the research work done using DEM has been given by Deen, van Sint Annaland, van der Hoef, and Kuipers (2007), Zhou, Kuang, Chu, and Yu (2010), and Zhu, Zhou, Yang, and Yu (2008).

There have been several attempts in modeling jetting fluidized beds. Semi-empirical correlations like fluidized bed grid zone model and Kunii–Levenspiel bubbling bed model were combined together and applied to a jetting fluidized bed coal gasifier by Kimura and Kojima (1992). They compared the local gas compositions in the grid zone to experimental results. Gas–solid flow behavior was investigated by Szafran and Kmiec (2004) for a spouted bed dryer with a draft tube. They utilized the Eulerian–Eulerian multi-fluid model to simulate the heat and mass transfer during the drying of grains in a spouted bed. A combined numerical and experimental approach to investigate the gas–solid flow dynamics was done by Zhang et al. (2002). They validated their numerical model with experimental results in a 2D spouted bed with a single central jet. Using the Eulerian method, Boemer, Qi, and Renz (1997) simulated bubble formation at a jet of a two dimensional fluidized bed. DEM was first used by Tsuji et al. (1993) to simulate a 2D jetting fluidized bed with the depth of the bed being equal to a single particle diameter. The fluidization pattern compared well with their experiments with a slight over prediction of the calculated pressure drop signal by DEM. 2D CFD–DEM simulations, with the bed depth being equal to a single particle diameter, have been conducted (Feng et al., 2004; Goldschmidt, Beetstra, & Kuipers, 2004; Hoomans, Kuipers, Briels, & van Swaaij, 1996; Hoomans, Kuipers, Mohd Salleh, Stein, & Seville, 2001; Mikami et al., 1998; Tsuji et al., 1993). Recently, flow regime maps were reported by van Buijtenen et al. (2011) using a DEM study, positron emission particle tracking and particle image velocimetry for double and triple spout fluidized beds. Numerical simulation and verification was done by Hong, Li, Cheng, and Zhang (1996) on a jetting fluidized bed.

In spite of the vast amount of data available on fluidized beds, numerical simulation of jetting fluidized bed with more than three jets has not been investigated extensively. In this work, we apply the DEM methodology to a 9-jet bed. Both two and three dimensional computations have been performed. A full scale 3D simulation of multiple jets comprising millions of particles has not been attempted previously. The computational results are compared with experimental measurements from Agarwal, Lattimer, Ekkad, and Vandsburger (2011), and Agarwal, Lattimer, Ekkad, and Vandsburger (2012). Particle fluxes, solid fraction and particle velocities near the jets are some of the quantities compared. The paper is structured as follows. The methodology section provides an overview of DEM followed by a brief outline of the experimental setup. The next two sections give the computational details and results, respectively, followed by a concluding section.

2. Methodology

DEM coupled with our in-house CFD code GenIDLEST (generalized incompressible direct and large eddy simulation of turbulence) has been used to numerically model the particle–fluid interactions in the jetting fluidized bed. Originally developed by Cundall and Strack (1979) for granular flows, Tsuji et al. (1993) used DEM for the first time to simulate 2D fluidized bed with a single jet. The volume averaged Navier–Stokes equations as derived by Anderson and Jackson (1967) are used for solving the fluid flow. Two different approaches can be used in DEM for resolving the particle–particle collisions. They are the hard sphere and soft sphere approaches. A soft sphere approach is considered in our analysis as it allows for multiple particle interactions which is more likely in a dense fluidized bed. A hard sphere technique only accounts for binary collisions and is better suited for dilute particle laden flows. The motion of each particle is tracked based on Newton's laws of motion as follows:

$$m_\mathrm{p}\frac{d\vec{v}_\mathrm{p}}{dt} = V_\mathrm{p}(\rho_\mathrm{p}-\rho_\mathrm{g})\vec{g} + \frac{V_\mathrm{p}\beta}{(1-\varepsilon)}(\vec{u}-\vec{v}_\mathrm{p}) + \vec{F}_\mathrm{p}, \tag{1}$$

$$I_\mathrm{p}\frac{d\vec{\omega}_\mathrm{p}}{dt} = \vec{T}_\mathrm{p}. \tag{2}$$

Integrating Eq. (1) in time advances the linear motion of the particle whereas Eq. (2) accounts for the rotational motion.

Each collision is modeled as particle overlap in the soft sphere model. A particular particle pair in collision has a normal and tangential linear spring and dashpot arrangement. The forces generated due to the overlap are calculated based on Hooke's law. The spring governs the deflection of the particle after collision, whereas the dashpot dissipates the energy of the particle, thus giving rise to inelastic collisions. In the tangential direction an additional sliding element is in series with the spring mass damper. The slider allows the particles to slide against each other as well, limiting the maximum magnitude of the tangential force. The normal and tangential forces due to the particle overlap are calculated based on the spring constants and damping coefficients. The following equations are used for the inter particle collisions:

$$\vec{f}_{\mathrm{n,pq}} = -k_n\vec{\delta}_{\mathrm{n,pq}} - \eta_\mathrm{n}\vec{v}_{\mathrm{n,pq}}, \tag{3}$$

$$\vec{f}_{\mathrm{t,pq}} = -k_\mathrm{t}\vec{\delta}_{\mathrm{t,pq}} - \eta_\mathrm{t}\vec{v}_{\mathrm{t,pq}}, \tag{4}$$

Please cite this article in press as: Deb, S., & Tafti, D.K. Two and three dimensional modeling of fluidized bed with multiple jets in a DEM–CFD framework. *Particuology* (2014), http://dx.doi.org/10.1016/j.partic.2014.04.005