Chemical Engineering Science 148 (2016) 108–125

Contents lists available at ScienceDirect

Chemical Engineering Science

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

Numerical investigation on the effect of draft plates on spouting stability and gas–solid characteristics in a spout-fluid bed

Shiliang Yang [a], Yuhao Sun [b], Liangqi Zhang [a], Ya Zhao [a], Jia Wei Chew [a,c,*]

[a] School of Chemical and Biomedical Engineering, Nanyang Technological University, 637459, Singapore
[b] Department of Engineering, University of Cambridge, Cambridge CB3 0FA, UK
[c] Singapore Membrane Technology Center, Nanyang Environment and Water Research Institute, Nanyang Technological University, 637141, Singapore

HIGHLIGHTS

- Spouting stability in a spout-fluid bed was investigated via CFD–DEM.
- Impact of draft plate with different lengths on improving spouting stability.
- Draft plates affect gas–solid hydrodynamics in spout, annulus and fountain.
- Cause of the dancing spout rooted in bubbles merging with spouting channel.
- Draft plate length optimized by balancing spout stability and solid circulation rate.

ARTICLE INFO

Article history:
Received 27 January 2016
Received in revised form
9 March 2016
Accepted 16 March 2016
Available online 26 March 2016

Keywords:
Spout-fluid bed
Spouting stability
Draft plate
CFD–DEM
Numerical simulation
Gas–solid hydrodynamics

ABSTRACT

Numerical simulation of dense gas–solid motion in a spout-fluid bed was carried out using the computational fluid dynamics coupled with discrete element method (CFD–DEM), in which the gas and solid motion are solved in the Eulerian and Lagrangian framework, respectively. After validating the simulated results with experimental data, the main cause of spouting instability was first identified, followed by evaluating the effect of draft plate length on spouting stability, pressure signals, and gas–solid hydrodynamics in the system. The results demonstrate that the onset of spout dancing, which is a type of spouting instability, is primarily due to the merging of the rising bubbles in the annulus with the spouting channel, which can be circumvented by the presence of draft plates. Increasing the draft plate length diminishes the mean pressure in the spouting inlet and the corresponding peak value of the power spectrum, and the correlation coefficient of pressure signals in the spout and background inlets. Regarding effect on the gas–solid hydrodynamics, a longer draft plate length leads to a more dilute upper spout, a higher spoutable height, higher voidage in the central fountain region, higher vertical gas flux (Fgz) and solid velocity (Usz) in the central axis, lower Fgz and Usz near the wall, and lower vertical solid flux (Fsz) overall. The optimization of the draft plate length depends on a balance between spout stability and solid circulation rate, since the former increases but the latter decreases with draft plate length.

1. Introduction

Compared with the stochastic motion of the solid phase in the bubbling fluidized bed (Chew et al., 2010; Hosseini et al., 2010), the more regular motion of the solid phase in the spouting apparatus makes the control of solid motion easier (Aguado et al., 2005; Hosseini et al., 2009). Thus, the spouting technology has been widely adopted in many physical and chemical processes, such as drying, granulation, pyrolysis and so on (Olazar et al., 2001, 2011). With the addition of background velocity to the traditional conical spouted bed, the resulting spout-fluid bed offers higher circulation and mixing rates of the solid phase, lower gas flow rates and the capacity for a wider operational range (Sutkar et al., 2013b). Consequently, the spout-fluid bed has been applied in tablet coating (Kfuri and Freitas, 2005), drying (Marmo, 2007; Zielinska and Markowski, 2007), coal/sawdust gasification (Lim et al., 1988; Thamavithya and Dutta, 2008; Xiao et al., 2006), among others.

In view of the advantageous features and widespread application of the spout-fluid bed, plenty of experimental efforts have

---

Abbreviations: 2-D, two-dimensional; 3-D, three-dimensional; CFD, computational fluid dynamics; DEM, discrete element method; MSA, mean square amplitude; TFM, two-fluid model

[*] Corresponding author at: School of Chemical and Biomedical Engineering, Nanyang Technological University, 637459, Singapore.
E-mail address: JChew@ntu.edu.sg (J.W. Chew).

http://dx.doi.org/10.1016/j.ces.2016.03.010
0009-2509/

been conducted to understand wide-ranging characteristics, such as voidage and solid velocity (Pianarosa et al., 2000), pressure fluctuation (Zhong and Zhang, 2005), flow regime (Link et al., 2005; Zhang and Tang, 2006), flow pattern and regime transition (Zhong et al., 2006b), particle mixing (Zhang et al., 2009, 2012) and circulation (Zhong et al., 2010), bed dynamics due to different collision properties (van Buijtenen et al., 2011), spout elevation (Jin et al., 2014; van Buijtenen et al., 2012), heat and mass transfer (Sutkar et al., 2015a), and behavior of a multiple-chamber system (van Buijtenen et al., 2011).

In addition to the experimental endeavors, advances in computational algorithm and hardware have made possible the numerical modeling of the spout-fluid bed. The two main approaches are the two-fluid model (TFM) and discrete element method coupled with computational fluid dynamics (CFD–DEM) (He et al., 2009, 2012; Hoef Van Der et al., 2008; Li et al., 2014; Zhu et al., 2007), between which the difference lies in the scale adopted to model the solid phase. On one hand, by means of TFM, the radial distribution of particle velocity and gas velocity showed similar tendencies, and the jet penetration depths were found to increase with pressure (Zhong et al., 2006c, 2007). On the other hand, the CFD–DEM approach has been more widely adopted to investigate the spout-fluid bed. Link et al. (2005, 2008) studied the flow regime in the spout-fluid bed by both CFD–DEM and experimental approaches, and characterized different flow regimes through the pressure fluctuations and the corresponding power spectra. Zhong et al. (2006a) and Zhang et al. (2010) found via CFD–DEM that, as spouting gas velocity increased, the percentage of particle–particle collision decreased (Zhong et al., 2006a), and the convective and shear mixing increased (Zhang et al., 2010). Goniva et al. (2012) investigated the effect of rolling friction of a spout-fluid bed using CFD–DEM to demonstrate that the comparison between numerical results with experimental data can be improved by applying a rolling friction model. Recently, by additionally considering heat transfer Patil et al. (2015) simulated via CFD–DEM a hot gas jet penetrating into a spout-fluid bed, and found that the hot gas jet caused bubble formation, which in turn induced particle circulation. Saidi et al. (2015) adopted CFD–DEM to conclude that bed thickness affected flow distribution. Using DEM, Yang et al. (2016) explored the influence of flow regime and operating parameters on solid dispersion in the spout-fluid bed and observed that increasing the total flow rate enhanced the solid dispersion intensity in all the directions.

In particular, an internal draft plate has been shown to be beneficial to the spout-fluid bed in terms of partitioning the wet spout and dry annulus zones to improve the performance of granulation and coating processes (Sutkar et al., 2015b). Experimentally, Berruti et al. (1988), Muir et al. (1990) and Nagashima et al. (2011) consistently demonstrated that the presence of draft plates significantly altered the hydrodynamics in the spout-fluid bed; for examples, the maximum possible spouting bed height appeared markedly greater (Berruti et al., 1988), the maximum attainable circulation rate occurred when the diameter of the jet was similar to that of the draft tube (Muir et al., 1990), and pressure fluctuations can be used to identify the flow regime (Nagashima et al., 2011). Recently, Sutkar et al. (2013a, 2013c, 2015b) constructed a flow regime map through experimental and numerical studies of the hydrodynamic characteristics of a spout-fluid bed with draft plates. Furthermore regarding the presence of draft plates, the same authors also experimentally investigated the heat transfer behavior when liquid was injected (Sutkar et al., 2015a), and more recently coupled heat and mass transfer via CFD–DEM to conclude that the draft plate allowed for concurrent wetting and drying contact zones (Sutkar et al., 2016).

All the above-mentioned experimental and numerical work all focused on the influence exerted by the draft plate on the gas and solid flow patterns in the spout-fluid bed. However, knowledge regarding the impact of the draft plate on spout stability, which is critical to the stable operation of the spout-fluid bed, and gas and solid behaviors remain absent. On spouting stability, a vertically submerged draft plate in a double slot-rectangular spouted bed (Chen, 2008; Yang et al., 2015) and a submerged plate in a prismatic spouted bed (Salikov et al., 2015) have both been shown to enhance stability. To bridge the gap in the knowledge base, the current effort is targeted at numerically investigating the effect of draft plates on spouting stability via CFD–DEM. As a first step, model validation was carried out by comparing the numerical results obtained in the absence of draft plates with the reported experimental data in the literature (van Buijtenen et al., 2011). Then, the onset of the spouting instability in the spout-fluid bed was identified, followed by comparatively studying the pressure drop and its power spectrum in the system with and without draft plates. Furthermore, the influence of the draft plate length on solid circulation rate and gas flow rate was discussed. More specifically, the objective of the current work is to understand the effect of draft plate length (L_d) on spouting stability, pressure signals, and gas–solid hydrodynamics (e.g., time-averaged distributions of velocity, voidage and flux) in the spout-fluid bed. Other parameters (e.g., gap between the draft plates) are expected to play roles, but are beyond the scope of the current study.

2. Governing equations for the two-phase flow

2.1. Governing equations for the gas motion

In the current work, the fluid involved is the gas phase. The gas motion is tracked in the Eulerian framework by taking into account the presence of the solid phase. The governing equation of gas motion with the incorporation of the voidage (ε_g) and momentum source terms can be formulated based on the continuity and Navier-Stokes equations:

$$\partial\left(\varepsilon_g\rho_g\right)/\partial t + \nabla\cdot(\varepsilon_g\rho_g\mathbf{u}_g) = 0 \tag{1}$$

$$\begin{aligned}\partial\left(\varepsilon_g\rho_g\mathbf{u}_g\right)/\partial t &+ \nabla\cdot(\varepsilon_g\rho_g\mathbf{u}_g\mathbf{u}_g)\\ &= -\varepsilon_g\nabla p_g - \textstyle\sum_{i=1}^{n}\mathbf{f}_{d,i}/\Delta V + \rho_g\varepsilon_g\mathbf{g} + \nabla\cdot(\varepsilon_g\tau_g)\end{aligned} \tag{2}$$

where t and $\mathbf{g}$ stand for the time and gravitational acceleration, respectively; ρ_g, $\mathbf{u}_g$ and p_g stand for the density, velocity vector and pressure of the gas phase, respectively. The viscous stress tensor (τ_g) and voidage (ε_g) are evaluated as

$$\tau_g = \mu_g[(\nabla\mathbf{u}_g) + (\nabla\mathbf{u}_g)^{-1}] + \left(\lambda_g - \frac{2}{3}\mu_g\right)(\nabla\cdot\mathbf{u}_g)\mathbf{I} \tag{3}$$

$$\varepsilon_g = 1 - \frac{\sum_{i=1}^{n}V_{i,t}}{\Delta V} \tag{4}$$

where λ_g, μ_g and $\mathbf{I}$ stand for the gas bulk viscosity, gas shear viscosity and second-order metric tensor, respectively; n, $V_{i,t}$ and ΔV represent the total particles in the cell evaluated, the total volume of particle i located in the current cell and the volume of the current cell, respectively. The momentum source for the current cell is represented by the summation of the drag force exerted on all the particles located in the current cell. The drag force $\mathbf{f}_{d,i}$ of the gas phase exerted on particle i can be formulated as

$$\mathbf{f}_{d,i} = \frac{V_i\beta_{gs}}{(1-\varepsilon_g)}(\mathbf{u}_g - \boldsymbol{v}_{p,i}) \tag{5}$$