Powder Technology 287 (2016) 51–60

Contents lists available at ScienceDirect

Powder Technology

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

Computational evaluation of depth effect on the hydrodynamics of slot-rectangular spouted bed

Kunzan Qiu, Chenshu Hu, Shiliang Yang, Kun Luo, Ke Zhang, Jianren Fan *

State Key Laboratory of Clean Energy Utilization, Zhejiang University, Hangzhou 310027, PR China

ARTICLE INFO

Article history:
Received 14 May 2015
Received in revised form 27 August 2015
Accepted 23 September 2015
Available online 26 September 2015

Keywords:
Computational fluid dynamics
Spouted bed
Discrete element method
Gas–solid flow
Depth effect

ABSTRACT

Computational simulation is a widely adopted method to study the spouted bed, or other industrial facilities. With the simulation problems becoming more and more complex, for example, higher solid particle numbers and higher demand to better describe the collisions between the particles, many researchers have contracted the third dimension of the bed in their simulation to ease the computational burden. In the present work, the gas–solid flow in a 3-D and quasi 3-D slot-rectangular spouted bed with same geometry parameters except the depth is numerically investigated with CFD-DEM to explore the depth effect on the hydrodynamics of the bed. The results demonstrate that: With bed depth increasing, the minimum spouting velocity and maximum pressure drop decreases initially and then remains a constant value. In the quasi 3-D bed, the flow pattern of steady spouting regime occurred in 3-D bed cannot form although a high superficial velocity is given, and the incoherent spouting state may cause the highest bed layer thickness to be twice that of the normal operation. In the spout region, the higher gas voidage in 3-D bed shows a larger spouting diameter than quasi 3-D bed. The time averaged flow rate in the spouting and motionless region of the quasi 3-D are obviously higher than that of the 3-D case.

© 2015 Elsevier B.V. All rights reserved.

1. Introduction

Spouted beds are widely used in industrial processes such as drying, coating, granulation, combustion and solid blending, which require very efficient mixing rate between granular materials [1,2]. Different from fluidized beds, the granular phase is spouted by the gas with a high velocity injected from a single nozzle, which causes the flow pattern in the beds consisted of three regions: "spout" region where upward gas–solid flow exists in the center, "annulus" region surrounding the spout where downward gas–solid flow exists and "fountain" region above the bed surface. For the industrial needs of efficiency and technological improvement, fundamental studies of granular materials in spouted bed are of great importance.

In studying industrial facilities containing gas–solid interactions with numerical simulation methods [3–8], discrete element model (DEM) [9–18] coupled with computational fluid dynamics (CFD) is a quite promising and well used method employed by many researchers [19,20]. Comparing to the two-fluid model (TFM), in which the solid phase is treated as continuous phase, the motion of the solid phase is calculated at the particle level in DEM. The model coupled with CFD (CFD-DEM) is more direct and fundamental, and it employs a continuous description for gas phase and can provide more real time information, such as the trajectories and transient forces acting on individual particles that can hardly been acquired by experimental methods.

CFD-DEM coupling method has been applied to explore the impact of operation parameters on the gas–solid hydrodynamics in the spouted bed. The effect of gas flow rate, the particle friction coefficient and the geometry basis angle were studied by means of CFD-DEM coupling method [21]. The influence of gas flow rate on the fluidizing behavior of solid phase has been investigated by Sailikov et al. [22]. The detailed particle dynamics with increasing gas velocity was described and a detailed regime map was provided. Due to the high calculation expense fulfilling DEM simulations, many researchers use simplified 2-D or quasi 3-D geometries to account for the three dimensional gas–solid behaviors in spouted bed, ignoring the influence of the third dimension of the bed. The simulations are usually applied on axisymmetric system or systems weakly influenced by the third dimension. The differences between 2-D and 3-D computational modeling have been investigated to better understand the limitation of such simplification. Xie et al. [23] applied continuum model to study the effect of 2-D and 3-D computational modeling. In comparison, the 2-D system could predict fluidized bed in bubbling regime well. Li et al. [24] reported a CFD simulation of circulating fluidized bed (CFB). It was found that 2-D simulation is hard to quantitatively predict the result in CFB. A pseudo 2-D rectangular spouted bed was reported by Hosseini [25]. The result shows that 3-D model predicts hydrodynamics with higher accuracy than 2-D model for spouted bed. In thin thickness apparatus, the effects of front and back wall are significant. And efforts have been taken to improve the prediction of bed hydrodynamics with 2-D simulation. A new model accounting for the frictional effect of front and back wall was proposed [26], which has showed significant improvement in numerical

---

* Corresponding author.
E-mail address: fanjr@zju.edu.cn (J. Fan).

http://dx.doi.org/10.1016/j.powtec.2015.09.034
0032-5910/© 2015 Elsevier B.V. All rights reserved.

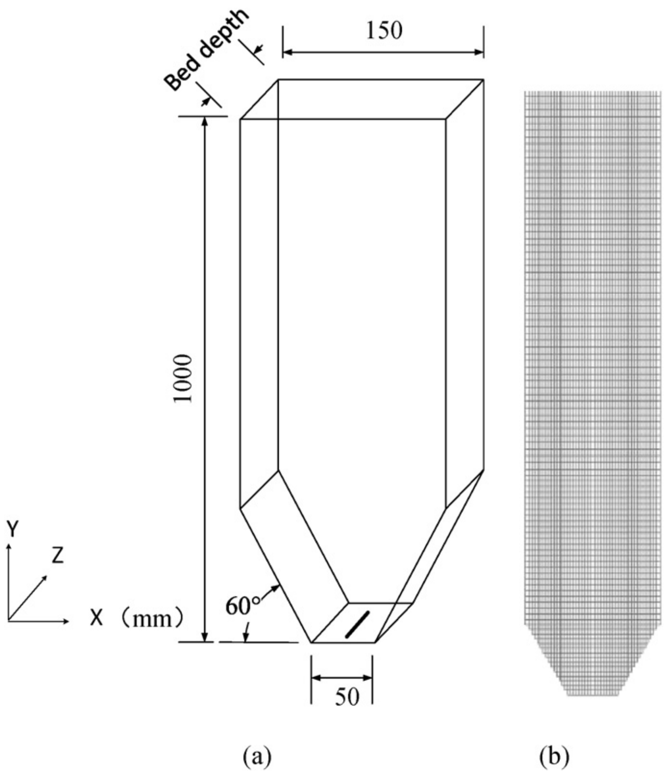

Fig. 1. (a) Schematic diagram of the slot-rectangular spouted bed. (b) Grid representation of the calculation domain [31].

prediction than traditional 2-D simulations. The influence of specularity on the fluidization behavior was studied by Altantzis et al. [27]. It was found that the appropriate value of specularity coefficient depends strongly on the superficial gas velocity. Zhao et.al [28] conducted a 2-D DEM simulation in which the bed width is 152 mm and the depth 15 mm. Freitas et.al reported in an experimental work [29] a bed with two relatively higher ratios of width to depth, in which the width is 150 mm, and the depths are 29 mm and 53 mm respectively. While in some situations, reasonable results cannot be acquired when the simplified geometries are adopted. In the study of Dogan et al. [30] and Freitas et al. [31], strong three dimensional characteristics arise in the slot-rectangular spouted bed relating to the gas–solid movement. Key parameters such as minimum spouting velocity, maximum spoutable bed height, maximum bed pressure drop, of the bed significantly change when keeping the Z-cross section shape constant while increasing the Z dimension thickness. Therefore, simulations carried with quasi 3-D and 3-D geometries may generate different results, which to date is seldom reported. The knowledge about the influence of bed depth is very important in the following two aspects. First, a quasi3-D fluidized bed with a thin bed depth suffers from the instability of discontinuous spouting regime [30]. Second, it provides important insights towards the scaling up of spouted bed.

In the present paper, to investigate the influences of the third dimension, i.e., the depth effects, on the gas–solid hydrodynamics, five spouted beds with same initial height of $H_b = 0.15$ m are simulated, with five different bed depths of 0.01 m, 0.04 m, 0.07 m, 0.10 m and 0.13 m, respectively. The evolution of minimum spouting velocity with increasing bed depth is obtained. The distinct flow regimes of incoherent spouting regime and internal jet with bubbling regime in quasi 3-D are described by the detailed startup procedures. Time averaged distributions of sold velocity, gas voidage and flow rate of solid phase in quasi 3-D and 3-D are compared to quantitatively analyze the differences between them.

2. Numerical schemes

2.1. Governing equations for gas motion

The fluid phase in this study, is treated as a continuum phase. The governing equations are the Navier–Stokes equations for the incompressible viscous Newton fluid motion in the spouted bed, and the volume fraction of gas is also taken into account, given by

$$\frac{\partial\left(\varepsilon_g\rho_g\right)}{\partial t}+\frac{\partial\left(\varepsilon_g\rho_g\mathbf{u}_i\right)}{\partial x_i}=0 \tag{1}$$

$$\begin{aligned}&\frac{\partial}{\partial t}\left(\varepsilon_g\rho_g\mathbf{u}_i\right)+\frac{\partial\left(\varepsilon_g\rho_g\mathbf{u}_i\mathbf{u}_j\right)}{\partial x_j}\\&=-\varepsilon_g\frac{\partial p}{\partial x_i}-\sum_{m=1}^{n}\frac{\mathbf{f}_{d,m}}{\Delta\mathrm{V}}+\rho_g\varepsilon_g\mathbf{g}+\frac{\partial}{\partial x_j}\left[\varepsilon_g(\mu+\mu_t)\left(\frac{\partial\mathbf{u}_j}{\partial x_i}+\frac{\partial\mathbf{u}_i}{\partial x_j}\right)\right]\end{aligned} \tag{2}$$

where ρ_g, p, $\mathbf{u}$, μ and μ_t are the density, the pressure, the velocity of fluid phase, the dynamic viscosity and the turbulent viscosity, respectively. $\mathbf{f}_{d,m}$ is the drag force exerted on particle m locating in current cell, and $\Delta\mathrm{V}$ is the volume of the computational cell. n is the total number of particles located in the current cell, and ε_g is the gas voidage and can be estimated as

$$\varepsilon_g=1-\frac{\sum_{i=1}^{n}V_{pi}}{\Delta\mathrm{V}} \tag{3}$$

where V_{pi} is the volume of particle i occupied by the current cell.

The turbulent viscosity is modeled with the k-ε turbulence model as

$$\mu_t=c_\mu\rho_g k^2/\varepsilon_t \tag{4}$$

where c_μ is a constant, $c_\mu = 0.09$. k and ε_t are, the turbulent kinetic energy and its dissipation rate, respectively.

The governing equations of turbulent kinetic energy and dissipation rate are given as

$$\begin{aligned}&\frac{\partial}{\partial t}\left(\varepsilon_g\rho_g k\right)+\frac{\partial\left(\varepsilon_g\rho_g k\mathbf{u}_j\right)}{\partial x_j}\\&=\frac{\partial}{\partial x_j}\left[\varepsilon_g\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+\varepsilon_g\mu_t\frac{\partial\mathbf{u}_i}{\partial x_j}\left(\frac{\partial\mathbf{u}_j}{\partial x_i}+\frac{\partial\mathbf{u}_i}{\partial x_j}\right)-\varepsilon_g\rho_g\varepsilon_t\end{aligned} \tag{5}$$

Table 1
Physical and numerical parameters for simulation.

Gas phase	
Inlet temperature, K	298
Viscosity, Pa·s	1.8×10^{-5}
Superficial gas velocity, m/s	1.6–0.2
Molecular weight, kg/mol	28.8
Pressure, atm	1
Particle	
Diameter, m	1.33×10^{-3}
Density, kg/m^3	2490
Spring constant, N/m	800
Particle-wall restitution coefficient	0.97
Interparticle restitution coefficient	0.97
Interparticle friction coefficient	0.3
Geometry	
Width, m	0.15
Depth, m	0.01, 0.04, 0.07, 0.10, 0.13
Height, m	1.00
Cells in x direction	49
Cells in y direction	192
Cells in z direction	2, 8, 14, 20, 26

Download English Version:

https://daneshyari.com/en/article/235158

Download Persian Version:

https://daneshyari.com/article/235158

Daneshyari.com