Chemical Engineering Science 93 (2013) 214–228

Contents lists available at SciVerse ScienceDirect

Chemical Engineering Science

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

Application of periodic boundary conditions to CFD-DEM simulation of gas–solid flow in pneumatic conveying

S.B. Kuang [a], K. Li [a], R.P. Zou [a], R.H. Pan [b], A.B. Yu [a,*]

[a] Laboratory for Simulation and Modelling of Particulate Systems, School of Materials Science and Engineering, The University of New South Wales, Sydney NSW 2052, Australia
[b] Longking Bulk Materials Science and Engineering Co., Ltd., Xiamen 361000, China

HIGHLIGHTS

- ▸ Horizontal pneumatic conveying is studied by CFD-DEM approach.
- ▸ Solid flowrate is controlled under periodic boundary conditions (PBC).
- ▸ A correlation is formulated to predict the start-up section length.
- ▸ Applicability of PBC and associated treatments is verified.

ARTICLE INFO

Article history:
Received 10 May 2012
Received in revised form
9 December 2012
Accepted 29 January 2013
Available online 8 February 2013

Keywords:
Multiphase flow
Pneumatic conveying
Granular materials
Periodic boundary conditions
Mathematical modeling
Simulation

ABSTRACT

This paper presents a numerical study of horizontal pneumatic conveying by the combined approach of computational fluid dynamics (CFD) and discrete element method (DEM), with special reference to the use of periodic boundary condition (PBC) for computational efficiency. A new iterative method is proposed to generally control the solid flowrate simulated in a CFD-DEM model with PBC. The characteristics of the flows in the start-up section are analyzed in detail and compared with those in the well-developed flow section for different flow regimes. On this basis, two semi-theoretical correlations are formulated to respectively predict the start-up section length and the relation between particle number and solid flowrate. The applicability of PBC to the CFD-DEM modeling of pneumatic conveying, as well as the two correlations, is verified by comparing the measured and calculated results under different conditions.

1. Introduction

Pneumatic conveying has increasingly been used in industries to transport bulk materials from one place to another (Mills, 2004). Its pipeline generally consists of start-up and established flow sections. In the start-up section, particles may form slugs/dunes or deposits, and experience acceleration before reaching a macroscopical equilibrium state in the established flow. This acceleration leads to a significant extra gas pressure loss (Hinkle, 1953; Rose and Duckworth, 1961; Kmiec and Leschonski, 1984) and different heat transfer behaviors from that in the established flow zone (Li and Mason, 2000; Li et al., 2003b; Narimatsu et al., 2007). Proper control of gas and solid behaviors in the start-up section can bring significant benefits to energy efficiency, product quality, and system stability by, for example, swirler (Li and Tomita, 1996, 1998, 2000), dune model (Rinoshika and Suzuki, 2010; Yan et al., 2012) and pulsed air knife (Wypych, 1995; Molerus, 1996). Therefore, the start-up section can be a determining factor of system performance, especially when pneumatic conveying is short in distance.

In the past, various theoretical and experimental efforts have been made to study the behaviors of gas and particles in the start-up section, leading to different theoretical and semi-theoretical correlations for the calculation of pressure drop and acceleration length mainly for dilute-phase pneumatic conveying (Hinkle, 1953; Rose and Duckworth, 1969; Yang and Keairns, 1976; Enick et al., 1987; Pinho, 1999, 2001). On the other hand, some one-dimensional hydrodynamic models have also been developed to study the flows in the start-up section (Kmiec and Leschonski, 1987; Littman et al., 1993; Grbavcic et al., 1997; Dzido et al., 2002). However, those studies were limited to low solid loadings due to the lack of general constitutive or closure relations for

---

[*] Corresponding author. Tel.: +61 2 93854429; fax: +61 2 93855856.
E-mail address: a.yu@unsw.edu.au (A.B. Yu).

0009-2509/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.ces.2013.01.055

particle–particle interaction in such a model. Generally speaking, the study of start-up section in pneumatic conveying is rather limited compared to those for the established flow section. This results in problems. For example, there are few data or correlations about the pressure drop and acceleration length for design purpose (Rizk, 1986). Because of the lack of knowledge of the flow in the start-up section in some experimental studies, the established flow zone had to be identified based on the pressure drop (Narimatsu and Ferreira, 2001; Rao et al., 2001), with no guarantee that particles really reached their equilibrium state (Kmiec and Leschonski, 1987). This problem may also exist in the numerical studies of pneumatic conveying.

In recent years, the combined approach of computational fluid dynamics (CFD) and discrete element method (DEM) has been widely accepted as an effective tool to study pneumatic conveying and other particle–fluid systems (Tsuji, 2007; Zhu et al., 2007, 2008; Van Der Hoef et al., 2008). Different CFD-DEM models have been developed and used to study various aspects in pneumatic conveying, as recently discussed by Kuang et al. (2008a, 2009, 2011) on different topics. In general, the previous CFD-DEM studies of pneumatic conveying are focused on established flows carried out by two different methods. One considers a long (up to 15 m) pipeline consisting of start-up and established flow sections. It requires high computational efforts, and at this stage of development, it cannot be applied to industrial pneumatic conveying where the pipelines may expand several miles. Another uses periodic boundary condition (PBC) in simulation to consider a short (0.5–2 m) pipe that represents the established-flow section which is often much longer than the start-up section. It is computationally efficient and thus has been widely adopted in the previous studies of pneumatic conveying (Tsuji et al., 1992; Kawaguchi et al., 2000; Yamamoto et al., 2001; Fraige and Langston, 2006; Lim et al., 2006a, 2006b; Mcnamara et al., 2006; Kuang et al., 2008a; Zhang et al., 2010; Zhou et al., 2010; Hilton and Cleary, 2011; Rao et al., 2011; Stratton and Wensrich, 2011). There are different problems associated with the application of PBC to pneumatic conveying, which are focused in this study. For example, the PBC treatment in the modeling of practical pneumatic conveying requires a method to exclude the start-up section from pipelines simulated, which is not yet available. Furthermore, a CFD-DEM model with PBC has to pre-set particle number and cannot generally control the solid flowrate simulated as a given value, due to the lack of correlation between particle number and solid flowrate (Kuang et al., 2008a; Kuang and Yu, 2011). Moreover, although PBC has been used in the numerical study of pneumatic conveying for some years, the results generated for a PBC pipe have not been seriously tested against those for a non-PBC pipe. To date, the method of selecting pipe length for a simulation with PBC is not very clear.

This paper presents a numerical study of horizontal pneumatic conveyer by means of a three-dimensional CFD-DEM model facilitated with or without PBC, aiming to understand the flows in the start-up section and overcome the aforementioned problems associated with the PBC application. It is organized as follows. The numerical model is first introduced, followed by a detailed study of the flow characteristics in the start-up section. Then, based on the results predicted, two correlations are formulated to respectively predict the boundary between start-up and established-flow sections, and the relation between solid flowrate and particle number. Finally, the applicability of PBC to CFD-DEM studies of pneumatic conveying is demonstrated by representative examples.

2. Simulation method

The CFD-DEM model used here has been discussed in detail in the previous study of horizontal pneumatic conveying (Kuang and Yu, 2011). A comprehensive discussion of the model formulation can also be found in the publication of Zhou et al. (2010). For brevity, therefore, we only describe the key features of the model below. However, the new development related to PBC will be detailed.

2.1. Governing equations for particle flow

The solid phase is treated as a discrete phase described by DEM, where the translational and rotational motions of a particle can be respectively described by the following equations:

$$m_i \frac{d\mathbf{v}_i}{dt} = \mathbf{f}_{pgf,i} + \mathbf{f}_{drag,i} + \sum_{j=1}^{k_i} (\mathbf{f}_{c,ij} + \mathbf{f}_{d,ij}) + m_i \mathbf{g} \tag{1}$$

and

$$I_i \frac{d\boldsymbol{\omega}_i}{dt} = \sum_{j=1}^{k_i} (\mathbf{T}_{t,ij} + \mathbf{T}_{r,ij}) \tag{2}$$

where m_i, I_i, $\boldsymbol{v}_i$, and $\boldsymbol{\omega}_i$ are the mass, moment of inertia, translational and angular velocities of particle i, respectively. According to Zhou et al. (2010), the main forces involved in the particle–fluid flow modeling are: (1) the pressure gradient force, given by $\mathbf{f}_{pgf,i} = -\nabla P V_i$, where P and V_i are the fluid pressure and the volume of particle i, respectively; (2) the fluid drag force, calculated by $\mathbf{f}_{drag,i} = \mathbf{f}_{drag0,i} \varepsilon_{f,i}^{-\chi}$, where $\boldsymbol{f}_{drag0,i}$ and ε_f are the fluid drag force on particle i in the absence of other particles and the local porosity for the particle respectively (Di Felice, 1994); (3) the gravitational force, $m_i\mathbf{g}$; and (4) the inter-particle forces between particles i and j, which include the elastic contact force, $\boldsymbol{f}_{c,ij}$ and viscous contact damping force, $\boldsymbol{f}_{d,ij}$. Trial simulations indicate that other particle–fluid forces, such as virtual mass force, lift force and viscous force, could be ignored in this work. The torque acting on particle i due to particle j includes two components. One arises from the tangential forces given by $\boldsymbol{T}_{t,ij} = \boldsymbol{R}_{i,j} \times (\boldsymbol{f}_{ct,ij} + \boldsymbol{f}_{dt,ij})$, where $\boldsymbol{R}_{i,j}$ is a vector from the center of mass to the contact point, and another is the rolling friction torque given by $\mathbf{T}_{r,ij} = \mu_{r,ij} d_i |\mathbf{f}_{n,ij}| \widehat{\boldsymbol{\omega}}_{t,ij}$, where $\mu_{r,ij}$ is the (dimensionless) rolling friction coefficient and d_i is the particle diameter (Zhou et al., 1999). The second torque is attributed to the elastic hysteresis loss and viscous dissipation in relation to particle–particle or particle–wall contacts, and it causes the decay in the relative rotational motion of particles. For viscoelastic material, it is recently reported that $\mu_{r,ij}$ can be evaluated as a function of materials properties (Zheng et al., 2011). For a particle undergoing multiple interactions, the individual interaction forces and torques are summed over the k_i particles in contact with particle i. The inter-particle and particle–wall forces are calculated based on the non-linear models commonly used in DEM (Zhu et al., 2007).

2.2. Governing equations for gas flow

The gas flow is treated as a continuous phase and modeled in a similar way to the one in the conventional two-fluid modeling. As discussed by Zhou et al. (2010), there are three different formulations available in the literature. The present work is based on the so-called original Model B formulation. Thus, its governing equations are the conservation of mass and momentum in terms of local mean variables over a computational cell, given by

$$\frac{\partial(\rho_f \varepsilon_f)}{\partial t} + \nabla \cdot (\rho_f \varepsilon_f \mathbf{u}) = 0 \tag{3}$$

and

$$\frac{\partial(\rho_f \varepsilon_f \mathbf{u})}{\partial t} + \nabla \cdot (\rho_f \varepsilon_f \mathbf{u}\mathbf{u}) = -\nabla P - \mathbf{F}_{p\text{-}f} + \nabla \cdot (\varepsilon_f \boldsymbol{\tau}) + \rho_f \varepsilon_f \mathbf{g} \tag{4}$$