Powder Technology 329 (2018) 241–251

Contents lists available at ScienceDirect

Powder Technology

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

# Computational investigation of particle flow characteristics in pressurised dense phase pneumatic conveying systems

Shanshan Sun [a], Zhulin Yuan [a,*], Zhengbiao Peng [b,*], Behdad Moghtaderi [b], Elham Doroodchi [b,c]

[a] Key Laboratory of Energy Thermal Conversion and Control of Ministry of Education, Southeast University, Nanjing 210096, China
[b] Priority Research Centre for Frontier Energy Technologies & Utilisation, The University of Newcastle, Australia
[c] Priority Research Centre for Advanced Particle Processing and Transport, The University of Newcastle, Australia

ARTICLE INFO

*Article history:*
Received 25 August 2017
Received in revised form 20 January 2018
Accepted 26 January 2018
Available online 31 January 2018

*Keywords:*
Pneumatic conveying
Dense phase
Numerical simulation
Bend
Flow characteristics
Solid concentration gradient

ABSTRACT

A new model for predicting particle flow characteristics in dense phase pneumatic conveying systems is presented. The domain of the solid phase is evenly divided by discrete grids, based upon which particle collision forces are solved. Specifically, the collision forces inside a grid are described based on the Lagrangian framework as a function of particle flow dynamics and the local solid concentration, whilst collision forces driving particles to flow amongst grids are calculated based on the Eulerian framework as the shear stress forces induced by the solid concentration gradient. The model was applied to solve the particle flow through a dense phase pneumatic conveying pipe with a 90° horizontal-to-vertical bend. Good agreements on the pressure drop through the horizontal and the bend sections were achieved between the prediction results and the experimental data. Effects of key parameters including superficial gas velocity, bend radius and particle size on the particle flow characteristics in terms of solid flow pattern, solid concentration distribution and fluctuation, and particle velocity distribution have been investigated. The results showed that the typical particle flow patterns, i.e. stratified flow, dune flow and slug flow, were sequentially observed when decreasing the superficial gas velocity or the bend radius, or increasing the particle size. The fluctuation intensity of solid concentration that is often seen as an indication of particle flow stability was substantially higher in the bend compared to those in straight sections. The model is simple yet proves efficient and capable of accurately predicting the particle flow characteristics in the dense phase pneumatic flow systems.

## 1. Introduction

Pneumatic conveying is a common transport system for bulk material which has been applied successfully in chemical (soap powders, detergents), food (sugar, flour), cosmetics (talc, face powder) or energy (coal and ash) industries [1]. The main advantages of pneumatic conveying over mechanical conveying are their enclosed nature, flexibility and easy automation. There are two basic types of pneumatic conveying systems: dilute phase (i.e. suspension flow with solid fraction <10%) and dense phase where the predominant flow mechanism is a non-suspension mode of flow. Compared to dilute phase systems, dense phase systems require much lower conveying velocities, thus lower power consumption in transportation and more importantly, are free of operational problems such as particle attrition and erosive wear of pipeline [1,2]. However, the dense phase pneumatic conveying systems are often intricate and feature complex gas-solid flow characteristics, e.g., intense pressure fluctuation and unsteady gas-solid flow hydrodynamics, both intimately dependent on the particle flow behaviour in the system. Therefore, a good understanding and the accurate prediction of the particle flow characteristics are pivotal for the design and operation of material handling and transportation systems that involve the dense phase pneumatic conveying process.

Studies have been conducted in the past decades, yet mainly focusing on the resistance performances of the pneumatic conveying system e.g., [3–5]. Experimental approach has been proven difficult due to instrumental and technological limitations regarding collection of data associated with individual particles in the bends. In contrast, numerical modelling proves to be a powerful and promising tool as it provides very detailed information on the local characteristics of the particulate flow. Indeed, significant attempts have been directed with the aid of computational fluid dynamics (CFD) models towards gaining a good understanding of particle flow characteristics through the bends in pneumatic conveying systems. Levy and Mason [6] conducted the pioneering CFD modelling work using the PHOENICS software (CHAM Ltd. UK) and studied the effect of a bend on the spatial distribution of solid particles as well as the phenomenon of segregation in the straight section in a dilute pneumatic conveying system. Despite the simpleness of their CFD model, the authors [6] found that the paths taken by the particles after the bend were strongly dependent upon their sizes. Akilli et al. [7] conducted numerical simulations using the commercial CFD

---

* Corresponding author.
*E-mail addresses:* zlyuan@seu.edu.cn (Z. Yuan), zhengbiao.peng@newcastle.edu.au (Z. Peng).

https://doi.org/10.1016/j.powtec.2018.01.078
0032-5910/

package (CFX) and investigated the dispersion characteristics of particles in a horizontal pipe after a 90° vertical-to-horizontal bend and observed the strong rope behaviour created by the bend which disintegrated within an axial distance of 10 pipe diameter. Meanwhile, the same research group [8] conducted CFD simulations using the Lagrangian particle-source-in-cell method to simulate the particle flow and investigated the non-uniformities of solid distribution that developed in the lean phase upward flow in the vertical pipe following a horizontal-to-vertical bend. Their results indicated a continuous rope-like structure formed within the bend which disintegrated into large discontinuous clusters at downstream location. Chu and Yu [9] combined their code of discrete element method (DEM) for the solid phase with the commercial software package Fluent (ANSYS) for the gas phase to investigate the typical features of the gas-solid flow in pneumatic conveying bends including roping, particle segregation, and particle velocity reduction. It was revealed that not only gas-solid interaction but also particle-particle interaction contributed to the dispersion of a rope [9]. Laín and Sommerfeld [10] compared different numerical models that were developed based on the single-phase flow, two-way coupling and four-way coupling (i.e. with the inclusion of inter-particle collisions) algorithms, and investigated the gas-solid flow pattern in the bend pipe. Good agreement between the simulation and the experimental data was achieved by the authors [10] only when the inter-particle collisions were considered in the model. Further, Kruggel-Emden and Oschmann [11] investigated the rope formation and dispersion for non-spherical particles including cubes, octahedrons, pyramids, plates and icosahedrons using a CFD-DEM approach. Their results showed that the differences in pressure drop, particle velocity distribution, rope dispersion and particle-particle, particle-wall and particle-fluid forces were strongly dependent on particle shape.

However, the majority of the above numerical studies were conducted for conditions of low pressure, or otherwise in dilute phase pneumatic conveying systems. The findings reported in those studies might not be valid to the pressurised dense phase pneumatic conveying systems. Generally speaking, a good understanding of the particle flow characteristics through the bends in terms of particle flow pattern, local solid concentration distribution and fluctuation, and particle momentum variation in pressurised dense phase pneumatic conveying systems is insufficient.

Moreover, with respect to the mathematical models available for simulating multiphase flow processes, models for solving the gas phase flow (e.g., finite element methods, controlled volume methods, Lattice Boltzmann methods) are relatively well-established. However, models for solving the particle flow are still confronted with challenges related to prediction accuracy and computational efficiency. Models that have emerged in the literature describing the solid phase could be grouped into two main frameworks: Eulerian and Lagrangian [12–14]. On the Eulerian Framework, the solid phase is treated as an interpenetrating continuum phase and has a separate but the similar set of governing equations with the fluid phase. Particle viscosity and pressure are solved based on statistical analysis, e.g. kinetic theory of granular flow (KTGF) derived from the Maxwellian velocity distribution of particles. Model based on the Eulerian framework is computationally preferable and can simulate a large number of particles for processes in industrial fluidised bed reactors [15–18]. However, the treatment of the solid phase as a continuum disobeys the discrete nature of solid particles and more importantly, the Eulerian models cannot provide information at particle-scale which is of utmost importance to studies of the particle flow behaviour in bends. Model developed on the Lagrangian framework, e.g., DEM and Direct Simulation of Monte-Carlo (DSMC) method, on the other hand, solves the motion of individual particles directly. A wealth of information on each particle (e.g., position, velocity, forces and diffusivity) can be obtained at any time instant of the flow [19–23]. Nevertheless, simulations using DEM is highly computationally demanding or provide unrealistic predictions results using DSMC that detects the collisions by probability. Therefore, there is a need for an alternative model that could overcome the shortcomings of the existing models for solving the particle flow.

In this study, a new model is presented for solving the flow of solid particles in a pressurised dense phase pneumatic conveying systems. The model is developed by combining the concepts of Lagrangian and Eulerian approaches. Particle flow characteristics in terms of solid flow pattern, solid concentration distribution and fluctuation, and particle flow velocity through a pneumatic conveying pipe with a 90° horizontal-to-vertical (HV) bend have been investigated. The key system parameters including superficial gas velocity, bend radius and particle size in determining the particle flow behaviour have been extensively examined in determining the particle flow behaviour. The model is validated by comparing the prediction results of pressure drop through both the horizontal and the bend sections against the experimental data published in the literature.

## 2. Mathematical model

### 2.1. Governing equations

The fluid flow is governed by the Navier-Stokes equations and the continuity equation, which can be readily found in the literature related to CFD simulations e.g., [24]. For brevity, the details of these equations are not given in the present study so as to specifically focus on the description of the new model for solving the motion of the solid phase.

In a dense fluid-solid flow, a single particle is interacting with neighbouring particles, surrounding fluid and computational domain boundaries. In general, the total force acting on a particle comprises collision contact forces, fluid forces and a gravitational force. The equations for the translational motion of particle *i* is,

$$m_i \frac{d\mathbf{v}_i}{dt} = \mathbf{f}_{c,i} + \mathbf{f}_{f,i} + m_i\mathbf{g} \tag{1}$$

where $m_i$ and $\mathbf{v}_i$ are the mass and velocity of particle *i*, respectively. $\mathbf{f}_c$ and $\mathbf{f}_f$ denote the collision forces and fluid forces acting on particle *i*, respectively. The interaction forces between a particle and boundary walls can be calculated by treating the wall as an infinite sphere, on which the model of particle–particle interactions can be applied. Governing equations for solving $\mathbf{f}_c$ and $\mathbf{f}_f$ are detailed as follows.

#### 2.1.1. Particle–particle interaction forces

Stratification has been reported as one of the most common phenomena that could occur in the dense phase pneumatic conveying process [25,26]. It thus seems plausible to describe the particle-particle interactions using a number of discrete grids that evenly divide the computational domain of the solid phase. Within a grid, the solution of particle-particle interaction forces is implemented based on the Lagrangian framework, whilst particle-particle interactions amongst these grids are calculated based on the Eulerian framework.

*2.1.1.1. Particle–particle interaction forces in a grid.* Based on the energy conservation law, collisions amongst particles in a grid incur the loss of momentum of particles, which can be expressed by,

$$v_{i,j} = a_{u,i} v_{i,j}^{(0)} \tag{2}$$

where $v_{i,j}^{(0)}$ and $v_{i,j}$ are the velocities of particle *j* in the grid *i* before and after collision. $\alpha_{u,i}$ is the bulk coefficient of restitution of particles in grid *i*. Subscript *u* represents the *x*, *y* or *z* direction. If no external energy is supplied to the system, all particles will eventually become still after the kinetic energy of particle is depleted due to energy dissipation during collisions.