Powder Technology 298 (2016) 30–41

Contents lists available at ScienceDirect

Powder Technology

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

A novel particle attrition model for conveying systems

Avi Uzi *, Haim Kalman, Avi Levy

Department of Mechanical Engineering, Ben-Gurion University of the Negev, P.O. Box 653, 84105 Beersheba, Israel

ARTICLE INFO

Article history:
Received 24 December 2015
Accepted 10 May 2016
Available online 12 May 2016

Keywords:
CFD-DEM
Particle attrition
Pneumatic conveying
One-dimensional modeling
Comminution processes

ABSTRACT

The ability to forecast how a conveying system will operate prior to its construction is a major issue for both industry and research. A trustworthy prediction can assist in the design of a conveying system and save efforts and resources spent in trial and error. In the present study, a novel model is presented for predicting the size reduction of particles coupled with the flow field in pipeline conveying. The model consists of two parts: a one-dimensional two-phase model that calculates the flow field and a one-dimensional breakage algorithm that accounts for particle collisions and breakage. The breakage algorithm exploits both empirical comminution functions defining the particle characteristics and machine functions (impact velocity and collision frequency) defining the system behavior, which are developed from CFD-DEM simulations. In the breakage model, the particles are described as individual discrete entities within a representative mass. This approach facilitates the use of low computational resources in the prediction of particle size reduction in conveying systems, thereby making it especially applicable for long-range industrial pipelines. Furthermore, in this method, it is possible to take into account particle fatigue and attrition not only in straight pipe sections but also in bends. In the current study, our new breakage model was implemented in a dilute-phase pneumatic conveying system. The machine functions developed for this system showed the following clear tendencies: Finer particles collide with the pipe wall at higher impact velocities and higher frequencies, while particle-particle interactions occur more frequently for coarser particles. Furthermore, it was found that breakage due to particle-particle interactions was significant inside bend elements, thereby implying that these collisions cannot be neglected. The model predictions of particle attrition were in good agreement, with a mean deviation of 5.8%, with the breakage results obtained previously by our group in dilute-phase pneumatic conveying experiments.

1. Introduction

Particle attrition during pneumatic conveying constitutes a major problem in a wide variety of industrial applications. When particles are transported pneumatically through a pipeline, they collide with each other and with the walls of the pipe, which may lead to particle breakage. In addition, particles may slide along the bottom of the pipeline, thereby being subjected to abrasive wear. These actions lead to particle degradation and hence to undesirable changes in material characteristics and product quality.

The attrition of particles during pneumatic conveying has been investigated experimentally in a large number of studies [1–11]. However, all are limited by the fact that the only way to experimentally examine particle attrition for a specific system is to compare the particle size distribution at the inlet to that at the outlet. Since particle attrition is governed by a vast number of parameters, it is extremely difficult to empirically correlate the particle size distribution at the end of the pipeline with that at the inlet, even for a large data set. For this reason, in the design of a new conveying line, there are no definitive 'rules' for obtaining the optimum operating conditions (pressure drop, air and particle mass flow rates, pipe diameter, number of bends, etc.) that will give the desired particle size distribution at the pipe outlet. One way to overcome this problem is to apply computational methods to predict the outcome product, but due to the complexity of the available models and the need for expensive computational resources, only a few researchers have implemented particle breakage in their simulations.

In the discrete element method (DEM) approach, particle breakage can be simulated by one of three different models [12]. Nonetheless, of these, only the 'fragments spawning' model of Brosh et al. [13] has been implemented to examine particle breakage in pneumatic conveying [14]. In this model, computational fluid dynamics (CFD)-DEM simulations are combined with the four following cumulative empirical comminution functions, which define the characteristics of the particles as a function of external loading: 1) the strength distribution to describe the compression force that would cause a particle to break [15]; 2) the breakage function, which is produced by horizontal impact system, to

---

Abbreviations: CFD, computational fluid dynamics; DEM, discrete element method; ODBA, one-dimensional breakage algorithm; SLR, solids loading ratio.

* Corresponding author.

E-mail addresses: uziav@post.bgu.ac.il (A. Uzi), hkalman@bgu.ac.il (H. Kalman), avi@bgu.ac.il (A. Levy).

http://dx.doi.org/10.1016/j.powtec.2016.05.014

describe the sizes of the fragments after a breakage event [16]; 3) the equivalence function to describe the relationship between the impact velocity and the equivalent compression load that would cause the same breakage probability as that if the particles had collided with a particular impact velocity [17]; and 4) the fatigue function [18] to take into account the weakening of the particle strength due to repeated loading. By implementing these functions within DEM simulations it becomes possible to determine whether a discrete particle will or will not break in a particular collision. Therefore, in this method, the fatigue and breakage for every discrete particle are calculated within the 3D domain. This tool, while effective, has the major drawback of a high computational cost, since the number of particles increases as the particles break into finer segments.

An alternate approach is to adopt a macroscopic model for two-phase flow and to simultaneously predict particle breakage, albeit with reduced accuracy but with the advantage of far less computation time. Such a degradation model was proposed by Chapelle et al. [10] and implemented on a large-scale dilute-phase pneumatic conveying line [19]. In this degradation model, a breakage matrix was derived from single impact tests and then implemented on every impact event in the simulation. The zero-dimensional flow model used by Chapelle et al. [19] was based the assumption that all particles travel at the same velocity (and therefore impact at the same velocity). Furthermore, the collisions were assumed to take place perpendicular to the pipe wall and only inside bends, thereby neglecting particle-particle collisions, collisions in straight pipe sections, and fatigue phenomena. There is thus a need for a model that will address these drawbacks.

In the present study, a new method is presented for calculating particle attrition due to impact events under steady-state flow conditions in conveying pipeline systems. The model consists of two parts: a one-dimensional two-phase flow model for calculating the fluid dynamics [20], and the one-dimensional breakage algorithm (ODBA) developed in this study for determining particle collisions and breakage with respect to the flow field.

In the ODBA, the methodology of Kalman et al. [21] is adopted for combining probability comminution functions (strength distribution, breakage, equivalence and fatigue functions) as the particle characteristic tool with the dynamics of the particles inside the system, namely, the ‘system behavior tool.’ The system behavior tool in the previously mentioned fragments spawning model is a 3D CFD-DEM simulation that describes the explicit motion of every discrete particle in a 3D domain. In contrast, the ODBA utilizes ‘machine functions’ as the system behavior tool. The machine functions are the collision frequency function, which describes the distribution of time between each collision, and the collision velocity function, which defines the distribution of the collision impact velocity. These functions must be known or developed prior to applying the ODBA.

The impact velocity function is used as the input for the comminution functions that determine whether a particle will break or weaken, and the collision frequency function is used to locate the positions of the collisions. These functions were developed in the present study for a particular experimental system [9] for investigating dilute phase pneumatic conveying of potash particles at several superficial gas velocities. The new method facilitates the calculation of the particle size distribution at any location along the axial direction of the pipeline.

In CFD-DEM breakage simulations, every simulated case provides a standalone result for the particle size distribution, which is accurate only for those specific simulation conditions. In principle, this approach facilitates a better classification of the attrition phenomena than the ODBA, since it can characterize every particle's collision event in a 3D domain. Nevertheless, it is impossible to apply this method in practice for long-range conveying systems and for process optimization because of the high computational cost. To overcome this drawback, the ODBA was developed as a fast calculation method for predicting attrition in conveying pipelines. Note that for developing the machine functions required for the ODBA, CFD-DEM simulations were conducted without taking particle breakage into account, which speeds up the simulation time in comparison with full CFD-DEM breakage simulations. Thus, in the present study, a set of CFD-DEM simulations was conducted for different conditions, the collision data was recorded, and correlations for the velocity and collisions frequency distributions were developed. In addition, the newly developed model was validated by comparing the model predictions with experimental results for conveying potash [9].

2. Model description

The purpose of the proposed model is to describe changes in particle size distribution along the pipeline, without conducting computationally expensive 3D CFD-DEM breakage simulations. Particle attrition is strongly affected by the carrier fluid flow, the particle concentration, and the geometry of the pipeline. The developed concept facilitates the prediction of the particle size distribution under different flow conditions in the following manner: The two-phase gas-solid flow field is calculated by a two-phase one-dimensional model. The ODBA is used for calculating the changes in particle size distribution within the flow field, utilizing, firstly, empirical comminution functions to obtain the particle characteristics [15–18] and, secondly, the machine functions as a system behavior tool to describe the dynamics of the particles inside the system. As mentioned above, the machine functions are the collision frequency function, which describes the distribution of time between each collision, and the collision velocity function, which defines the distribution of the collision impact velocity.

In the present study, we developed the above two functions by applying 3D CFD-DEM simulations. In the DEM, particles are modeled as discrete entities, with the realization of Newton's Second Law on each particle for calculating the particles' motion. The DEM solver is coupled with an Eulerian phase solver (ANSYS Fluent™ in this study), which calculates the fluid dynamics. Therefore, this method allows us to capture every individual collision event throughout the simulation. Furthermore, the methodology of capturing the collision data allows us to store this data for subsequent statistical analysis with respect to the geometry of the system, the physical properties of the carrier fluid, and the material being conveyed. For each different case simulated, the ODBA will produce a range of results that can then be applied to produce a general correlation for the two machine functions. Since the development of general machine functions requires a large database, in the specific system that was simulated to verify the model in the present study, the machine functions were developed separately for each simulated case.

2.1. One-dimensional two-phase flow model

The dynamics of the two-phase flow was predicted using a one-dimensional model with the following assumptions: 1) The flow along the pipeline is steady state. 2) Air is an ideal gas. 3) The solid holdup of the particles is equivalent to the particles' cross-sectional coverage area. 4) The particles have a uniform size distribution in the pipe cross section. 5) The particles are fully suspended, resulting in a homogenous concentration in the pipe cross section. 6) The kinetic energy loss of the particles due to bend collisions can be obtained from CFD-DEM simulations and implemented into the one-dimensional model. 7) In dilute phase pneumatic conveying, since the greater part of the pressure drop due to bends in the pipeline is caused by a change in direction of the gas phase, pressure drops due to bends are assumed for the gas phase alone:

$$\Delta P = 0.5 \rho_g \nu_g^2 \mathrm{K} \tag{1}$$

where, ν_g is the gas velocity before the bends, ρ_g is the gas density, and K is the energy loss factor, which is taken as 0.25, typically for R/D = 1. It should, however, be noted that the acceleration of the particles after the