Powder Technology 268 (2014) 219–236

Contents lists available at ScienceDirect

Powder Technology

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

Numerical study of rope formation and dispersion of non-spherical particles during pneumatic conveying in a pipe bend

H. Kruggel-Emden *, T. Oschmann

Ruhr-Universitaet Bochum, Universitaetsstrasse 150, D-44780 Bochum, Germany

ARTICLE INFO

Article history:
Received 3 June 2014
Received in revised form 27 July 2014
Accepted 14 August 2014
Available online 23 August 2014

Keywords:
Pneumatic conveying
Pipe bend
Simulation
DEM–CFD-approach
Non-spherical particles
Rope formation

ABSTRACT

In pneumatic conveying bends are used to interconnect vertical and horizontal pipe sections. During conveying bends are known to be related to several characteristic flow phenomena. Experimentally, their investigation turns out to be difficult which favors the use of numerical approaches instead. Detailed investigations become possible by relying on Euler–Lagrange methods. Here, particularly combined Discrete Element Method and Computational Fluid Dynamics (DEM–CFD) approaches allow the transient description of the particle/fluid interaction. So far, only spherical particles have been considered by the DEM–CFD during pneumatic conveying with very few exceptions, e.g. [1], as DEM–CFD frameworks capable of representing non-spherical particles are not yet widely established. There is an ongoing discussion on proposed frameworks regarding applicable particle/fluid force models and the way of the particle shape representation. Due to these unresolved questions conveying of non-spherical particles through pipe bends involving phenomena like rope formation and dispersion has not been considered with numerical approaches such as the DEM–CFD, yet. Therefore, as many bulk solids involve complex shaped particles, rope formation and dispersion are investigated for non-spherical particles by a DEM–CFD approach for the first time. Exemplarily, cubes, octahedrons, pyramids, plates and icosahedrons are addressed. A DEM–CFD framework is developed which allows the modeling of arbitrary shaped particles. To underline the validity of the approach, results are benchmarked against established correlations which are available for certain particle shapes. Obtained results indicate differences in the pressure drop, particle velocity distribution, rope dispersion and the particle/particle, particle/wall and particle/fluid forces which are strongly dependent on particle shape.

1. Introduction

Pneumatic conveying of bulk solids is important to industries varying from agricultural, chemical, energy conversion, plastics, food, pharmaceutical to minerals processing [2]. The advantages of pneumatic conveying are a high level of safety, low operation costs, flexibility in terms of layout, simple installation/automation and low maintenance. The disadvantages are equipment and bulk solids wear and a large energy demand. Pneumatic conveying is often performed in horizontally or vertically aligned pipe sections which are linked by bends. Among all the equipment pipe bends are least investigated and are not well understood despite their simplicity [3]. Typical problems related to bends appear in a strong contribution to the overall pressure drop, possible product degradation, erosive wear and a not well understood and strongly product dependent rope formation and dispersion especially in industrial applications.

In order to increase the overall bend design as well as to understand the complex flow patterns, mathematical modeling is a powerful tool which relies on different modeling frameworks. In Euler–Euler approaches [4] both fluid and particle phase are modeled as two interpenetrating continua based on the kinetic theory of granular flow. Alternatively, approaches are established by a Euler–Lagrange description where the fluid phase is modeled as a continuum and the particles as discrete entities. Depending on the required level of detail, the particle sizes and concentrations, collisions among particles as well as a two way coupling between particles and fluid phase are neglected in the most simplified frameworks. More advanced models include a two way coupling between particles and fluid phase and represent particle/particle collisions based on stochastic models [5] or by deterministically tracking individual particles including their collisions over time. The latter approach is known as the combined Discrete Element Method and Computational Fluid dynamics (DEM–CFD) approach [6] and allows a transient description of both fluid and solid phases. Details on the state of the art of the outlined methods and to which extent they are capable of modeling particle/fluid systems comprising of non-spherical particles are discussed in the following.

Generally, Euler–Euler models are preferred in applied research or for process modeling, as their computational load is lower than that of Euler–Lagrange models. However, for detailed investigations they are not applicable because they do not correctly represent segregation phenomena [7]. Particles have to be small with regard to the system

---

* Corresponding author. Tel.: +49 234 32 27362; fax: +49 234 32 14 227.
E-mail address: kruggel-emden@leat.rub.de (H. Kruggel-Emden).

http://dx.doi.org/10.1016/j.powtec.2014.08.033

size to fulfill the continuum assumption and shape cannot be correctly represented (limitation to spheres), although progress has been recently made for the modeling of polydisperse particle systems [8,9]. Even for spherical particles a large drawback of Euler–Euler methods is their limited capability to represent the rotational degree of freedom influenced by possible tangential particle interactions. Additionally, multi-particle interactions are not modeled for [7] which is a weakness especially in dense systems.

Despite the known drawbacks, Euler–Euler frameworks built on the kinetic theory of granular flow [4] were considered firstly by Eskin [10] for the modeling of pneumatic conveying within a 1D model. Despite its simplicity comparisons with experimental data and results obtained from Lagrangian simulations proved its general applicability. Dense pneumatic conveying in vertical [11] and horizontal directions [12] was simulated by a 3D Euler–Euler framework for coal dust by Pu et al. at elevated pressure. Results indicate a reasonable representation of pressure drop and solids concentrations. Plug flow in a pipeline enlargement was addressed by a Euler–Euler model by McGlinchey et al. [13]. Just recently Behera et al. proposed a 1D model for dense phase [14] and for dilute conveying of very fine particles [15].

Euler–Lagrange frameworks allow for a more precise modeling of particle fluid systems compared to Euler–Euler models as particles are tracked as discrete entities. The fluid phase is often only considered in steady-state. Proposed modeling frameworks usually focus on spherical particles e.g. [16–20], with the exception of [21] (details are given below). Often particle/particle collisions are neglected which limits the applicability of the frameworks to dilute flow situations. In that case particle/particle collisions are modeled for, stochastic approaches are often used which are accurate [5] but limited in their application to spherical particle shapes. Euler–Lagrange frameworks therefore generally disqualify for setups involving non-spherical dense particle flow such as the one targeted in the investigation here.

Examples for Euler–Lagrange frameworks include works by Lain and Sommerfeld [21] who investigated steady-state pneumatic conveying of non-spherical particles in a horizontal channel. Due to dilute conditions particle/particle collisions were neglected and particle/wall collisions were modeled by a stochastic approach. For the drag force the particle asphericity was considered. Rotational motion which strongly influences drag forces (comp. [22]) as well as lift forces was not modeled for. Probably due to the dilute conditions, obtained velocity profiles were in reasonable agreement with experimental investigations. Based on an extended framework Lain and Sommerfeld [16] later investigated spherical particles within horizontal flow for different wall roughnesses. The extended framework [16] additionally considered particle/particle collisions, lift forces and the rotational particle motion and was also used for the investigation of horizontal flow in circular pipes [17] and ducts [18]. A detailed comparison of channel and pipe flow was performed by Lain and Sommerfeld [19] more recently. A detailed model similar to that of Lain and Sommerfeld [19] was developed and applied by Aletto and Breuer [20] for the investigation of secondary flow characteristics formed during horizontal conveying.

The combination of the Discrete Element Method coupled to Computational Fluid dynamics (CFD) is a special, refined form of Euler–Lagrangian frameworks and always relies on a two-way coupling as well as a deterministic collision description [6]. The coupled DEM–CFD usually resolves both fluid and particle flow transiently and allows for the inclusion of arbitrarily shaped particles [23] and therefore perfectly qualifies for the simulation setup considered in this investigation. A large drawback of the DEM–CFD method currently is that frameworks capable of representing non-spherical particles reliably are not yet established. Only a handful of studies have been performed involving the DEM–CFD in the context of complex particle shapes e.g. [23–27] and only one investigation addressed pneumatic conveying (channel flow) so far [1].

In the context of pneumatic conveying the DEM–CFD was firstly applied by Tsuji et al. [28]. Horizontal plug flow of spherical particles was investigated and results were compared to experimental investigations. The particle phase was considered in 3D; the fluid phase was modeled in steady-state by a 1D-formulation. The obtained numerical results were identified as reasonable with regard to the wave-like motion of the flow boundary. Stationary layer thickness and plug flow velocity matched experimental results. More than 10 years after the pioneering work by Tsuji et al. [28], Han et al. [29] used a 2D transient DEM–CFD model to address attrition during transport of spherical particles in a conveying system consisting of horizontal and vertical pipe elements. Ouyang et al. [30] applied a 2D hard sphere model and coupled it to CFD for the simulation of gas/solid flow in a vertical pipe. Axial as well as radial pressure drops in dense phase plugs were investigated experimentally and numerically by Mc Glinchey et al. [31] based on a 2D DEM–CFD model. Based on a 3D DEM–CFD-model involving spheres Lim et al. [32] reproduced different flow patterns such as dispersed and plug flow for vertical conveying and homogenous, stratified, moving dunes and plug flow for horizontal conveying as reported from experimental investigations. Lim focused on voidage waves in hydraulic conveying for narrow pipes [33]. Similar to the approach of Tsuji et al. [28], Fraige and Langston [34] investigated horizontal conveying of spherical particles employing a 3D DEM model coupled to a steady-state 1D CFD approach. Flow patterns reported experimentally were quantitatively reproduced. Plug flow in vertical conveying and slug flow in horizontal conveying were investigated by Strauss et al. [35,36]. Inclined pneumatic conveying was investigated numerically and experimentally by Zhang et al. [37]. Experimental matched numerical

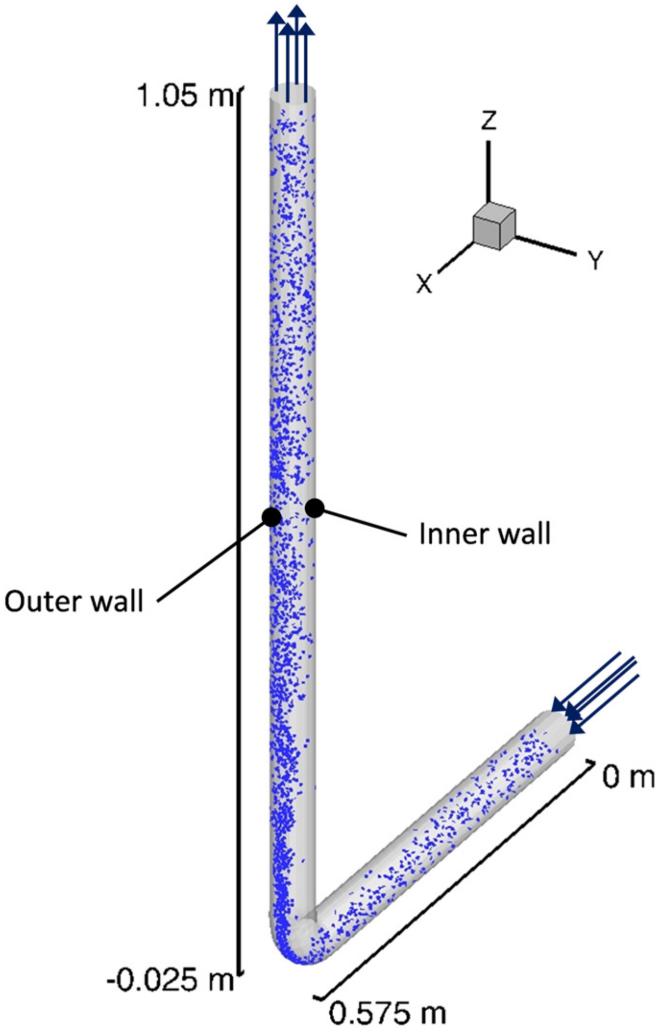

Fig. 1. Schematic diagram of the simulation setup.