Powder Technology 327 (2018) 456–466

Contents lists available at ScienceDirect

Powder Technology

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

On the application of an Eulerian granular model towards dilute phase pneumatic conveying

J. Heng [a,b], T.H. New [a,*], P.A. Wilson [b]

[a] *School of Mechanical and Aerospace Engineering, Nanyang Technological University, 50 Nanyang Avenue, 639798, Singapore*
[b] *Faculty of Engineering and the Environment, University of Southampton, Southampton, SO167QF, England*

ARTICLE INFO

Article history:
Received 16 June 2017
Received in revised form 18 December 2017
Accepted 22 December 2017
Available online 3 January 2018

Keywords:
Pneumatic conveying
Two-phase flow
Eulerian granular model
Numerical simulation

ABSTRACT

The present study considered the application of a multiphase model with Eulerian approach for the solids phase in dilute-phase conveying, where the results are compared against previously published experimental results based on 42 μm nominal diameter glass particles. In particular, the Favre-Averaged Drag turbulent dispersion model is studied where it is found to have greater effects on the particle concentration distribution as compared to the gas phase velocity. While certain discrepancies are observed between simulations and published experimental data, the flow characteristics are adequately captured after addressing the underlying cause of inaccuracies. Inaccuracies in the particle concentration distributions along a vertical pipe section result from the difficulty in capturing the transitional zone where the particle rope starts to disperse. On the other hand, particle diameter variations underpin the mismatches along a horizontal pipe section. Interestingly, increasing the particle diameter leads to the successful capturing of the particle concentration distribution along the horizontal pipe section. The accuracy of employing an Eulerian approach for solids phase is demonstrated, provided that effects due to the particle diameter are accounted for.

1. Introduction

The use of pneumatic conveying systems to transport bulk materials over extended distances is commonplace for many industries. They had been utilized in power generation industry for the transport of pulverized coal; food industry for the transport of powdery ingredients such as wheat, flour or sugar; and construction industry for the transport of cement powder, to name a few. It stems from the many benefits that a pneumatic system provides over conventional belt conveying [1]. As the different industries continue to look for ways to improve their conveying systems, either for safety concerns or for increasing throughput, pneumatic conveying has been studied for many decades. Numerical simulations, or computational fluid dynamics (CFD), have emerged over the years as a reliable approach to study the mechanisms of pneumatic conveying. Investigations of dense or dilute flow regimes such as particle-laden flows, two-phase or multiphase flows, and moving bed flows, etc., are feasible through numerical simulations. Dense and dilute phase conveying are terms usually used to identify under which regime the flow is classified. Dense phase conveying generally encompass flows where the material is transported as a distinct phase from its carrier gas at the bottom of the channel whereas dilute flows generally mean that the particulates are entrained in the carrier gas phase. More detailed discussions on these flow classifications can be found in [1–4].

Due to their complexities, experimental studies are still being conducted to study pneumatic conveying processes. Some examples will include work by Khan et al. [5] on the pick-up velocity of particles and de Moraes et al. [6] on head loss coefficient for pipe fittings, both of which deal with properties fundamental to pneumatic conveying processes. While experimental studies form a significant portion of pneumatic conveying investigations, the number of numerical simulation studies is increasing as well. Over the years, many researchers had come up with many different models catering to different conditions or areas of interest in the flow. Apart from direct numerical simulations (DNS) and solving the complete equations of motion for each individual particle, there is no one unified model to use for gas-particle flows that captures the intricate flow behaviour very accurately. In most cases, only the most appropriate modelling was applied for certain cases to reduce computational costs. These included the selection of either an Eulerian or Lagrangian approach of the solids phase based on the flow regime [7], a two-way or four-way coupling between the phases depending on the particle loading ratio [8], Stokes number [9], and various turbulent models, although some models have consistently shown higher accuracies over other models. Descriptions of these numerical models will be presented in greater detail in Section 2.

The most commonly investigated aspect within pneumatic conveying is the pressure drop within pipes. McGlinchey et al. [10] used the

---

* Corresponding author.
E-mail address: dthnew@ntu.edu.sg (T.H. New).

https://doi.org/10.1016/j.powtec.2017.12.069

Nomenclature

C_{TD}	Turbulent dispersion constant
d_s	Particle diameter
D_s	Dispersion scalar
e_{ss}	Coefficient of restitution
F_{td}	Turbulent dispersion force
g	Acceleration due to gravity
$g_{0,\,ss}$	Radial distribution function
I	Identity tensor
k_{Θ_s}	Diffusion coefficient
K_{pq}	Interphase momentum exchange coefficient
p	Pressure
R_{pq}	Interaction term between phase p and q
Re	Relative Reynolds number
t	Time
v	Velocity
α	Volume fraction
γ_{Θ_s}	Collisional dissipation of energy
Θ_s	Granular temperature
λ	Bulk viscosity
μ	Shear viscosity
ρ	Density
σ	Turbulent Prandtl number
τ	Stress tensor
ϕ	Internal angle of friction

Eulerian formulation and predicted pressure drops in 90° bends. Their results demonstrated qualitative agreement with experimental data. In another case, Makwana et al. [11] looked into the effects of dune formation on pressure drops. Dunes, formed at the bottom of the pipe during some conveying processes, were found numerically to induce large pressure fluctuations. In other aspects, Li et al. [12] predicted solid deposition characteristics using the Lagrangian formulation. As the Lagrangian model was used, only fairly large particles were modelled within their computations. Nevertheless, the study was able to shed light on the interaction between particles. They described how a particle plug would interact with the layer of deposited material left by the previous plug, and how the plug leaves some material behind as it traverses along the pipe. Hidayat and Rasmuson [13] investigated the gas-solid flow characteristics within a U-bend in attempt to optimize a pneumatic conveying dryer system. They detailed the effects of the bend radius on the distribution of particles and the gas velocity, as well as further concluding the importance of these two parameters on the efficiency of the dryer. Other applications include investigation of plug flows in step-up pipe arrangements [14], of very fine powders [15], of bend orientations [16] and of a combination of two bends [17]. Such investigations demonstrated the practical applications of numerical simulations in the understanding pneumatic conveying flow characteristics in different situations.

Consequently, there was a demand for higher accuracy levels in numerical simulation models. As researchers work towards developing more robust and accurate numerical models to study particle-laden flows, most of them have turned towards using a Lagrangian particle treatment for dilute flows [18]. One of the reasons for solving the complete equations of motion for the particles is to obtain highly accurate simulations of particle-laden flows. However, the primary downside of the Lagrangian model is that with current computing architectures, it is simply not feasible to scale up the computations to study real-life industrial applications due to computational costs [19]. For studies involving more complex geometries or higher solids loading ratios, the Eulerian scheme remains the only practical approach in terms of computational cost and time. However, one of the concerns surrounding the Eulerian scheme is its assumption of a continuum. This means that the actual dynamics of the flow starts to deviate from the assumptions as the particle phase gets more dilute, i.e. the distance between particles increases [2]. Van Deemter and van der Laan [20] performed the pioneering work of deriving the balance equations by integration methods. Anderson and Jackson [21] later on used the equations of motion for a particle, together with the Navier-Stokes equations for fluid motion, and came up with the set of governing equations that is still used today in many commercial computing software. However, many constitutive relations could not be obtained directly by their method. A large part of this initial gap has since been filled with models either derived with another method or by empirical correlations. These include, for example, the interface drag term [22–24], granular temperature [25] and solid shear stresses [26].

One of the closure relations that should be highlighted here is the turbulent dispersion model. Because the distribution of particles (even in areas with relatively low particle concentrations) has been shown to affect the flow structures of the gas phase [27–29], it is essential to accurately model the particle distributions. Turbulent dispersion is the mechanism for the particle phase to be transported from regions of high concentration to lower concentrations [30] and this is especially so after a pipe bend. Burns et al. [30] utilized Favre averaging of the interphase drag term to obtain the additional terms in the momentum equations that account for turbulent dispersion. The authors had named it the Favre Averaged Drag model. This model finds itself most commonly used in scenarios pertaining to gas-liquid flows (i.e. droplets or bubbly flows). Of the currently known publications that either referred to or utilized the Favre Averaged Drag (FAD) model in their numerical models, none had applications dealing with gas-solid dilute phase flows. This is despite the original authors concluding that the model is a generalization of other turbulent dispersion models and it should hold true for a wide range of multi-phase flow scenarios. Furthermore, to the best knowledge of the present authors, there were no known restrictions raised in the existing literature in terms of applying this particular turbulent dispersion model towards a dilute gas-solid flow.

This study primarily aims to demonstrate the ability of an Eulerian scheme in predicting the flow characteristics of a granular phase in a pneumatic pipe configuration featuring a 90° bend. Both the upstream and downstream portions of the pipe bend will be studied, with a focus on the particle distribution, as well as both the gas and particle velocities. As detailed above, the application of the FAD model towards a gas-solid multiphase flow remains scarce in the literature. Here, the authors intend to document the effects of the FAD model on a flow where one of the phases is a granular phase. Specifically, the empirical model constant will be varied to observe its effects and to select a suitable constant. The objective is to investigate if this model possesses the ability to be extended to gas-solid dilute flows. Lastly, the effects of the particle diameter used within the model will also be briefly touched upon.

2. Numerical model

The Eulerian scheme (also known as the Eulerian-Eulerian scheme in some cases) treats both the gas and solid phases separately. It incorporates a set of conservation equations for each phase and is interpenetrating. The gas is treated as the usual fluid phase by the numerical model. For the solid phase, the solid was treated as a continuum akin to the gas phase and is only differentiated by designating it as a granular phase. A granular phase has additional parameters and constitutive relations that are required to fully model the phase. One advantage of the Eulerian granular model is that it can be used for comparatively large computational domains with proportionally more solid particles. As opposed to the Lagrangian method where the equation of motion is solved for each and every particle within the system, the Eulerian granular model solves only one conservation equation for each defined solid phase. Therefore, the Eulerian granular model scales in complexity only