A Computational Study of the Various Flow Regimes in Pneumatic Conveying of Granular Materials

Eldin Wee Chuan Lim and Chi-Hwa Wang

Department of Chemical and Biomolecular Engineering, National University of Singapore, 4 Engineering Drive 4, Singapore 117576.

Prepared for presentation at the 2005 AIChE Annual Meeting

Cincinnati, Ohio, October 30 - November 4, 2005.

September, 2005

Introduction

Gas-solid systems are commonly encountered in the chemical and petrochemical, food and mineral processing and pharmaceutical industries. Their applications include fluid catalytic cracking, drying operations, mixing and granulation and the transport of granular material and fine powders through pipelines. In particular, the pneumatic transport of granular material is a common operation frequently employed to transport solid particles from one location to another. Some of the advantages associated with this method of solid transportation include relatively high levels of safety, low operational costs, flexibility of layout, ease of automation and installation and low maintenance requirements. On the other hand, one of the main disadvantages of pneumatic transport is the occurrence of attrition of the granular material, especially at high conveying velocities. This may result in severe degradation of product quality in certain industrial applications and possibly unpredictable changes in flow behaviors within the conveying pipelines. Depending on the system geometry, gas velocities and material properties of the solid particles to be transported, such transportation processes can take place in different modes usually referred to as dense or dilute-phase conveying. The former involves transportation of the solids as dense clusters or slugs and is usually the preferred method for handling solids which are sensitive to abrasion as shear and collisional forces arising within the solid material are generally lower. In comparison, the latter mode where particles are dispersed as a suspension in the gas is known to be a more stable mode with lower fluctuations and excursions in gas pressures.

Numerical modeling of pneumatic conveying and other gas-solid systems plays an important role in improving our understanding of such systems. One of the commonly used approaches to pneumatic conveying modeling is the Eulerian/Lagrangian method where particles are tracked in a Lagrangian frame of reference either individually or as groups with identical properties known as parcels.[1, 2] An alternative approach has been Computational Fluid Dynamics (CFD) with two-fluid continuum models to represent the gas and solid phases as two interpenetrating continua.[3] Further, the technique of particle dynamics simulation has also been widely used for investigations of granular and gas-solid systems. In particular, the Discrete Element Method (DEM) originally developed by Cundall and Strack[4] for describing the mechanical behavior of assemblies of discs and spheres, has been successfully applied by many research workers in various areas of engineering interests. Tsuji et al.[5] carried out numerical simulations of horizontal pneumatic conveying of solid particles using DEM and showed that particles moved in the form of plugs in the conveying pipe. Several research workers have also applied the approach of combining DEM with CFD to the simulation of two-dimensional fluidized beds.[6–10] Li and Mason[11] used the same approach to model heat transfer between gas, solid particles and pipe wall in a pneumatic conveying system. Han et al.[12] simulated the flow of salt particles through a dilute phase pneumatic conveying system to predict particle attrition and breakage.

An understanding of the differences in physics between the various flow regimes found in pneumatic conveying of granular material may be important to actual industrial or commercial applications with regards to the optimality of operation, ease of control and extent of damage inflicted on the solid particles as well as the conveying lines. Despite the large number of work reported on gas-solid systems, there have been relatively fewer attempts at modeling the various flow regimes in vertical and horizontal pneumatic conveying systems. The ability to predict the flow behaviors of both gas and solid phases during a typical pneumatic conveying operation or the modes in which the transportation would take place remains limited. As such, the objective of this study is to apply the technique of combining DEM with CFD to the numerical simulation of pneumatic conveying of granular material in both vertical and horizontal pipes. The emphasis has been on reproducing computationally the different types of solid flow patterns and behaviors observed experimentally under different operating conditions. In the following sections of this paper, the DEM and CFD models used and their methods of implementation will be described, and simulation results obtained will be compared with experimental observations reported in the literature.

Mathematical Model

Discrete Element Method

The translational and rotational motions of individual solid particles are governed by Newton's laws of motion:

$$m_i \frac{d\mathbf{v}_i}{dt} = \sum_{j=1}^{N} \left(\mathbf{f}_{c,ij} + \mathbf{f}_{d,ij}\right) + m_i \mathbf{g} + \mathbf{f}_{f,i} \quad (1)$$

$$I_i \frac{d\boldsymbol{\omega}_i}{dt} = \sum_{j=1}^{N} \mathbf{T}_{ij} \quad (2)$$

where m_i and $\mathbf{v}_i$ are the mass and velocity of particle i, N is the number of particles in contact with this particle, $\mathbf{f}_{c,ij}$ and $\mathbf{f}_{d,ij}$ are the contact and viscous contact damping forces respectively, $\mathbf{f}_{f,i}$ is the fluid drag force due to an interstitial fluid, I_i is the moment of inertia of particle i, $\boldsymbol{\omega}_i$ is its angular velocity and $\mathbf{T}_{ij}$ is the torque arising from contact forces which will cause the particle to rotate.

Contact and viscous contact damping forces have to be calculated using force-displacement models which relate such forces to the relative positions, velocities and angular velocities of the colliding particles. In the present work, the linear force-displacement model was implemented according to the following equations:

$$\mathbf{f}_{cn,ij} = -\kappa_{n,i} \boldsymbol{\delta}_{n,ij} \quad (3)$$

$$\mathbf{f}_{ct,ij} = -\kappa_{t,i} \boldsymbol{\delta}_{t,ij} \quad (4)$$

$$\mathbf{f}_{dn,ij} = -\eta_{n,i} \left(\mathbf{v}_r \cdot \mathbf{n}_i\right) \mathbf{n}_i \quad (5)$$

$$\mathbf{f}_{dt,ij} = -\eta_{t,i} \left[\left(\mathbf{v}_r \cdot \mathbf{t}_i\right) \mathbf{t}_i + \left(\boldsymbol{\omega}_i \times \mathbf{R}_i - \boldsymbol{\omega}_j \times \mathbf{R}_j\right)\right] \quad (6)$$

where $\mathbf{f}_{cn,ij}$, $\mathbf{f}_{dn,ij}$ and $\mathbf{f}_{ct,ij}$, $\mathbf{f}_{dt,ij}$ are the normal and tangential components of the contact and viscous contact damping forces respectively, $\kappa_{n,i}$, $\boldsymbol{\delta}_{n,ij}$, $\mathbf{n}_i$, $\eta_{n,i}$ and $\kappa_{t,i}$, $\boldsymbol{\delta}_{t,ij}$, $\mathbf{t}_i$, $\eta_{t,i}$ are the spring constants, displacements between particles, unit vectors and viscous contact damping coefficients in the normal and tangential directions respectively, $\mathbf{v}_r$ is the relative velocity between particles and $\mathbf{R}_i$ and $\mathbf{R}_j$ are the radius vector (from particle center to a contact point) for particles i and j respectively. If $\left|\mathbf{f}_{ct,ij}\right| > \left|\mathbf{f}_{cn,ij}\right| \tan\phi + c$ then ‘slippage’ between the two contacting surfaces is simulated by a Coulomb-type friction law, $\left|\mathbf{f}_{ct,ij}\right| = \left|\mathbf{f}_{cn,ij}\right| \tan\phi + c$ where tanϕ is analogous to the coefficient of friction and c is a measure of cohesion between the two contacting surfaces.

Fluid Drag Force

In a multiphase system such as the gas-solid pneumatic conveying system considered in this study, interactions between the two phases take the form of fluid drag forces on the solid particles exerted by the interstitial fluid and arise from velocity differences between the two phases. In this study, the model due to Di Felice[13] which is applicable over a wide range of particle Reynolds numbers was used for evaluating the fluid drag force term in Eq. (1). Following Xu et al.[14], the modified equations in this model are shown as follows:

$$\mathbf{f}_{f,i} = \mathbf{f}_{f0,i}\varepsilon_i^{-(\chi+1)} \tag{7}$$

$$\mathbf{f}_{f0,i} = 0.5c_{d0,i}\rho_f \pi R_i^2 \varepsilon_i^2 \left|\mathbf{u}_i - \mathbf{v}_i\right|\left(\mathbf{u}_i - \mathbf{v}_i\right) \tag{8}$$

$$\chi = 3.7 - 0.65\exp\left[-\frac{\left(1.5 - \log_{10} \mathrm{Re}_{p,i}\right)^2}{2}\right] \tag{9}$$

$$c_{d0,i} = \left(0.63 + \frac{4.8}{\mathrm{Re}_{p,i}^{0.5}}\right)^2 \tag{10}$$

$$\mathrm{Re}_{p,i} = \frac{2\rho_f R_i \varepsilon_i \left|\mathbf{u}_i - \mathbf{v}_i\right|}{\mu_f} \tag{11}$$

where $\mathbf{f}_{f0,i}$ is the fluid drag force on particle i in the absence of other particles, χ is an empirical parameter, ε_i is the local average porosity in the vicinity of particle i, $c_{d0,i}$ is the drag coefficient, $\mathrm{Re}_{p,i}$ is the Reynolds number based on particle diameter, ρ_f is the fluid density, μ_f is the fluid viscosity and $\mathbf{u}_i$ is the fluid velocity of the computational cell in which particle i is located.

Computational Fluid Dynamics

The motion of the continuum gas phase is governed by the Navier-Stokes equations with an additional source term in the momentum equation to represent the reaction force acting on the fluid by the particles:

$$\frac{\partial \varepsilon}{\partial t} + \nabla \cdot (\varepsilon \mathbf{u}) = 0 \tag{12}$$

$$\frac{\partial(\rho_f \varepsilon \mathbf{u})}{\partial t} + \nabla \cdot (\rho_f \varepsilon \mathbf{u}\mathbf{u}) = -\nabla P + \nabla \cdot (\mu_f \varepsilon \nabla \mathbf{u}) + \rho_f \varepsilon \mathbf{g} - \mathbf{F} \tag{13}$$

where $\mathbf{u}$ is the velocity vector, ε is the local average porosity, P is the fluid pressure and $\mathbf{F}$ is the source term due to fluid-particle interaction.

Simulation Conditions

The geometry of the pneumatic conveying system and type of particles used in the present simulations were based on the experimental work of Rao et al.[15] and Zhu et al.[16] (Table 1) so that a meaningful comparison between the simulation and experimental outputs can be made. The gas velocities considered in this study were in the ranges 14 m s^{-1} – 24 m s^{-1} and 10 m s^{-1} – 30 m s^{-1} for the vertical and horizontal pneumatic conveying simulations respectively because these would include all the flow regimes observed via Electrical Capacitance Tomography measurements by Rao et al.[15] and Zhu et al.[16] for the two systems. The numbers of particles used were 500, 1000, 1500 and 2000 corresponding to overall solid concentrations α, of 0.08, 0.16, 0.24 and 0.32 respectively where α is defined as the overall volume fraction of particles divided by the volume fraction of particles at maximum packing which is generally taken to be 0.64. The equivalent coefficient of restitution represented by the viscous contact damping coefficient selected for the present study was found by conducting a numerical experiment similar to that used by Xu and Yu.[7] Without loss of generality, the same values of the coefficients of friction and restitution as shown in Table 1 were used for both particle-particle and particle-wall interactions. The flow patterns from the numerical simulations were then compared qualitatively with the experimental observations of Rao et al.[15] and Zhu et al.[16]

Table 1. Material properties and system parameters

Shape of particles	Spherical
Type of particles	Polypropylene
Number of particles	500, 1000, 1500, 2000
Particle diameter, d	2.8×10^{-3} m
Particle density, ρ_p	1123 kg m^{-3}
Spring constant in force model, κ	5.0×10^{3} N m^{-1}
Viscous contact damping coefficient, η	0.35
Coefficient of restitution	0.1
Coefficient of friction	0.3
Cohesion, c	0.0
Coefficient of rolling friction, μ_r	5.0×10^{-5} m
Gas density, ρ_f	1.205 kg m^{-3}
Gas viscosity, μ_f	1.8×10^{-5} N s m^{-2}
Pipe diameter	0.04 m
Pipe length	1.0 m
Computational cell size	4 mm × 4 mm
Simulation time step, Δt	10^{-7} s

In all simulations performed, particles were first allowed to settle freely under gravity for 0.5 s and form a packing at the ‘bottom’ of a vertical pipe or a heap in a horizontal pipe before gas flow was initiated. Periodic boundary conditions were applied to the solid phase to simulate an open flow system while a uniform gas velocity profile was maintained at the inlet. Particles which were carried out of the conveying pipe by the flowing gas were simulated to re-enter from the inlet of the pipe with the same velocities and radial positions. The main advantage of this method was the possibility of simulating a very long conveying pipe using a significantly smaller computational domain which leads to more efficient utilization of computing resources.

Results and Discussion

Vertical pneumatic conveying

The combined CFD-DEM model described in this paper was first used for the numerical simulation of pneumatic conveying of granular material in a vertical pipe. From the simulation outputs obtained, two distinct types of flow regimes could be identified. When the solid concentration was 0.08 and the gas velocity was 14 m s^{-1}, particles were seen to be distributed throughout the entire length of the pipe. This kind of flow pattern is known generally as dispersed flow. When α was increased to 0.24 and 0.32, particles move in the form of a single large plug along the conveying pipe. Generally, it is known from previous experimental work reported in the literature that the dispersed flow regime is usually dominant at high gas velocities and low solid concentrations while the plug flow regime is observed otherwise.[16]

The solid concentration profiles for both flow regimes were obtained by dividing the space in the conveying pipe into long strips parallel to the length of the pipe and calculating the solid concentration within each strip. It was observed that this spatially averaged solid concentration profile became invariant with time after a sufficiently long simulation time, indicating the attainment of a fully developed flow state. Figures 1 and 2 show the solid concentration profile for the dispersed (α = 0.08) and plug (α = 0.32) flow regimes at various gas velocities respectively. It may be observed that, contrary to the name of the regime which can be made from the instantaneous snapshots of the simulation, the solid concentration profile for dispersed flow shows a symmetrical but non-uniform distribution with higher solid concentrations near the walls and a minimum near the center of the pipe. This may be due to the effects of inelastic collisions with the walls which leave particles lying in the vicinity of the walls. On the other hand, the solid concentration profile for the plug flow regime is uniform and flat across the section of the pipe. This is due to the fact that particles in such a regime are closely packed together into a single large plug which moves much like a rigid body along the conveying pipe. For each of these regimes studied, the solid concentration profiles do not seem to be significantly affected by the actual velocity of the gas used.

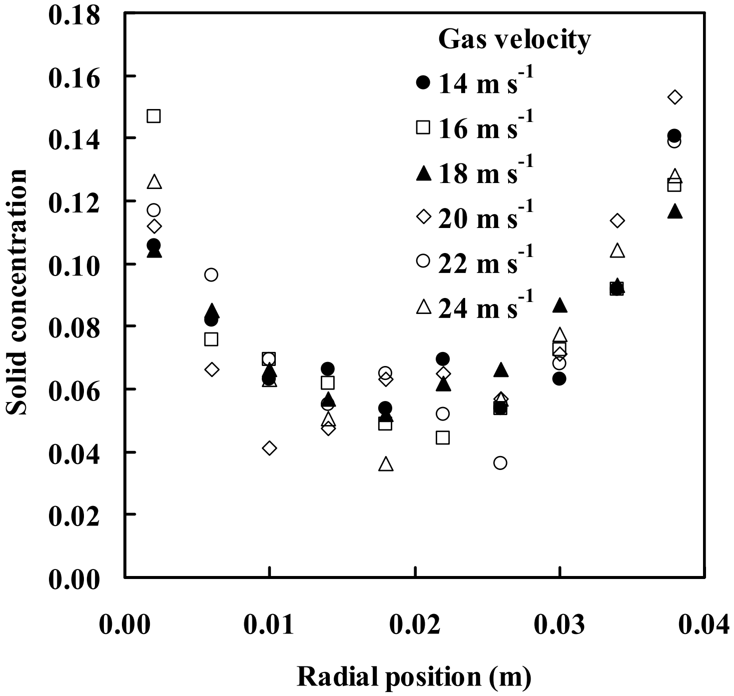

Figure 1 Solid concentration profile for the dispersed flow regime in vertical pneumatic conveying ($\alpha = 0.08$) at various gas velocities showing symmetry and minimum near the pipe center

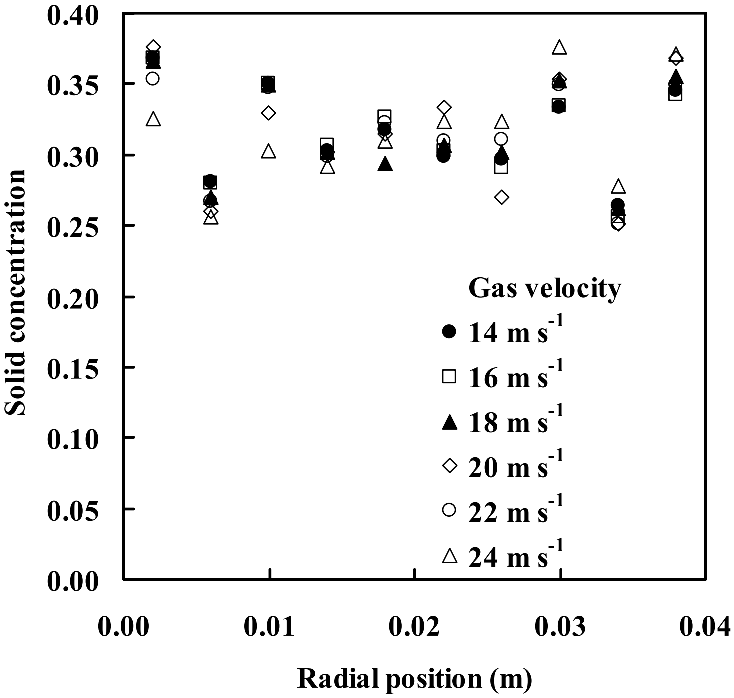

Figure 2 Solid concentration profile for the plug flow regime in vertical pneumatic conveying ($\alpha = 0.32$) at various gas velocities showing a flat distribution

Horizontal pneumatic conveying

The simulation outputs for horizontal pneumatic conveying showed a few other types of flow patterns in addition to those observed in vertical pneumatic conveying. These arise mainly due to gravitational effects which cause particles to settle towards the bottom wall of the conveying pipe. At a low solid concentration of 0.08 represented by 500 particles and conveying gas velocity of 10 m s^{-1}, the flow pattern observed resembles that of dispersed flow in vertical pneumatic conveying but due to the effects of gravitational settling as mentioned, a thin layer of particles is formed along the lower pipe wall. There exists a gradient in the concentration of particles in the radial direction with higher concentration of particles near the lower pipe wall and vice versa. This flow regime was also observed under similar operating conditions experimentally and is known as the stratified flow regime.[15] With $\alpha = 0.16$, the previously observed thin settled layer of particles became larger clusters which move along the lower wall by traction. The individual clusters do not seem to have a tendency to combine together nor be dispersed into suspension but remain quite stable throughout the entire simulation time. A large portion of the particles is still transported in suspension above these moving clusters. Following the experimental work of Rao et al.[15], this is referred to as the moving dunes flow regime. In contrast, at the highest solid concentrations of 0.24 and 0.32 considered in the present simulation using 1500 and 2000 particles respectively, particles tend to be transported in the form of a single large cluster reminiscent of plug flow in vertical pneumatic conveying. This may be a result of clustering of multiple adjacent moving dunes to form a stable large plug which spans the entire cross-section of the conveying pipe. This kind of flow pattern was similarly observed in physical experiments done at high solid concentrations and low gas velocities and was called the slug flow regime.

The solid concentration profiles for two representative regimes in horizontal pneumatic conveying, stratified flow ($\alpha = 0.08$) and slug flow ($\alpha = 0.32$), were similarly computed for a quantitative comparison of the effects of gravitational settling mentioned earlier on the resulting solid distribution. Figure 3 shows quantitatively that the solid concentration is higher near the bottom wall of the horizontal pipe when particles are conveyed in the stratified flow regime, corresponding to the qualitative observations made from the simulation snapshots seen previously. By comparison with the solid concentration profile for dispersed flow in vertical pneumatic conveying, the effect of gravitational settling has caused the profile to change from a symmetrical distribution to an unsymmetrical one. In contrast, the solid concentration profile for slug flow (Figure 4) is seen to be very similar to that for plug flow in vertical pneumatic conveying. This fits well with the fact that particles are carried in the form of a single large plug in both cases. When this occurs in a horizontal pipe, the effect of gravity does not alter the solid concentration profile to any significant extent.

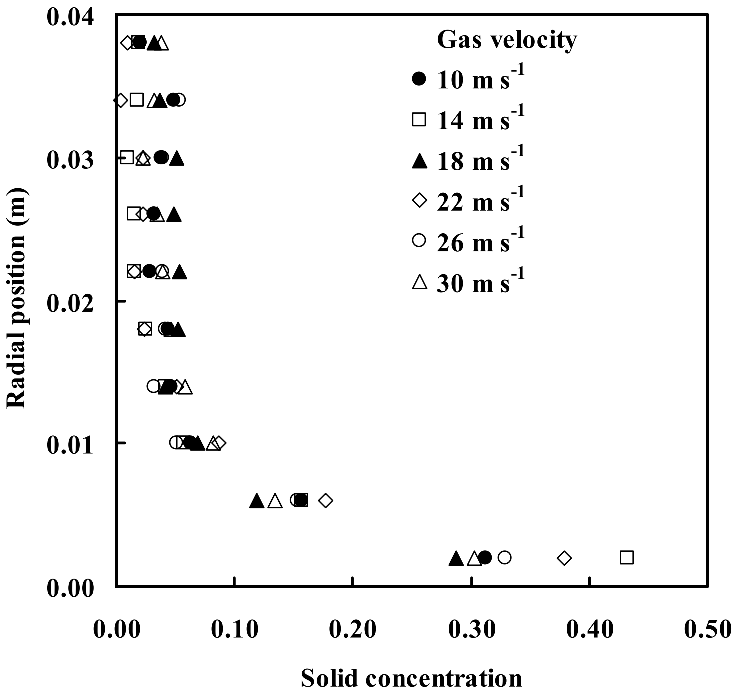

Figure 3 Solid concentration profile for the stratified flow regime in horizontal pneumatic conveying (α = 0.08) at various gas velocities showing non-symmetry and higher solid concentration near the bottom wall

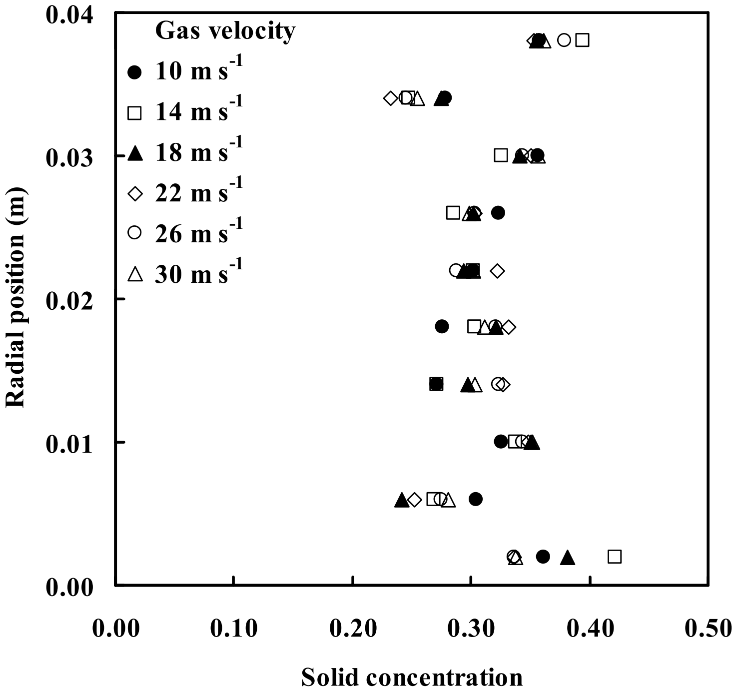

Figure 4 Solid concentration profile for the slug flow regime in horizontal pneumatic conveying (α = 0.32) at various gas velocities showing a flat distribution (Order of coordinates is different from Figure 11 to aid in visualization)

Phase diagrams

The different flow regimes in vertical and horizontal pneumatic conveying arising from the use of different operating conditions can be represented in the form of phase diagrams as shown in Figures 5 and 6 respectively. Dashed lines in the figures separate approximately the regions representing different flow regimes while dashed circles enclose regions where transition between two adjacent flow regimes might be taking place. In vertical pneumatic conveying, the dispersed flow regime is dominant at high gas velocities and low solid concentrations while the plug flow regime is dominant otherwise (Figure 5). This is also generally true for horizontal pneumatic conveying except at low gas velocities and solid concentrations where the effects of gravitational settling of particles result in the formation of the moving dunes and stratified flow regimes (Figure 6). Intermediate values of gas velocities where transitions between the moving dunes and homogeneous flow regimes (MD/H) and between the stratified and homogeneous flow regimes (S/H) are similarly approximated by regions enclosed within the dashed circles in Figure 6.

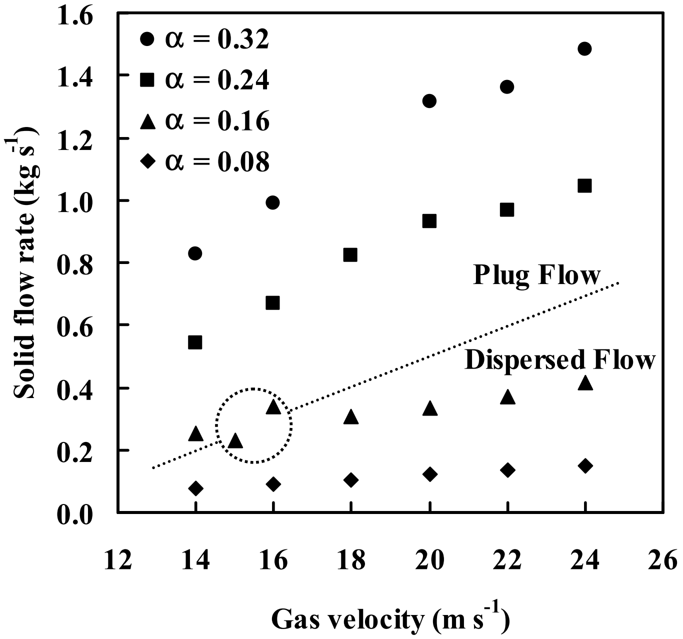

Figure 5 Phase diagram for vertical pneumatic conveying. Dashed lines separate approximately regions representing different flow regimes while dashed circles enclose regions where transition between two adjacent flow regimes might be taking place. The dispersed flow regime is dominant at high gas velocities and low solid concentrations while the plug flow regime is dominant otherwise.

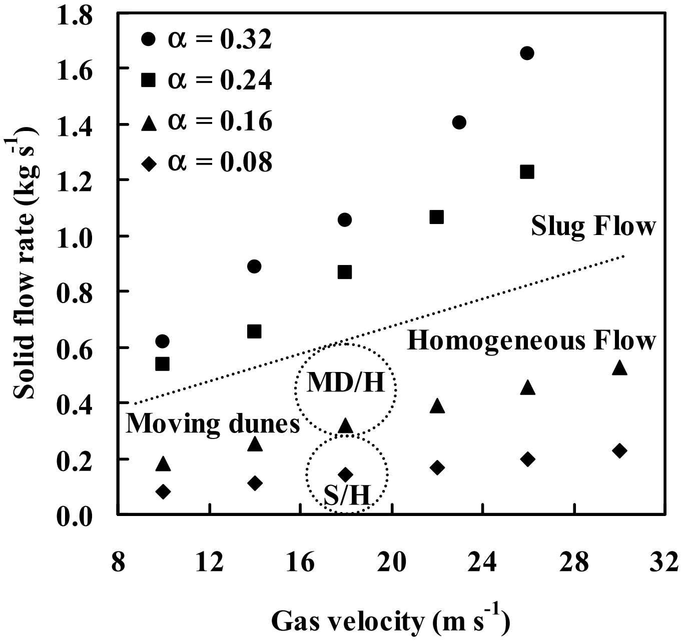

Figure 6 Phase diagram for horizontal pneumatic conveying. Homogeneous flow is dominant at high gas velocities and low solid concentrations. The effects of gravitational settling result in the formation of the moving dunes and stratified flow regimes at low gas velocities and solid concentrations. MD/H and S/H denote transitions between moving dunes and homogeneous flow and between stratified and homogeneous flow respectively.

Conclusions

The Discrete Element Method utilizing a linear spring-dashpot-friction slider force-displacement model was coupled with Computational Fluid Dynamics and used for the simulation of pneumatic conveying of granular material in both vertical and horizontal pipes in this study. The motions of solid particles and gas were obtained by time integration of Newton's second law of motion and the Navier-Stokes equations respectively. Fluid drag forces were calculated using a fluid-particle drag force model and also represented as a source term in the gas momentum equation to ensure satisfaction of Newton's third law between the two phases. The effects of rolling friction and collision dynamics have also been considered in the computational model developed.

The simulation results obtained were in good agreement with previously reported experimental observations in terms of the types of flow patterns arising at different operating conditions used. In vertical pneumatic conveying, particles tend to be dispersed throughout the pipe at high gas velocities and low solid concentrations. On the other hand, particles tend to cluster together and move in the form of a dense plug when gas velocities are low or solid concentrations are high. These flow patterns have been referred to as the dispersed and plug flow regimes respectively. The solid concentration profile for dispersed flow was observed to be symmetrical and with a minimum near the center of the pipe while that for plug flow was almost flat. In horizontal pneumatic conveying, the simulations also show the presence of homogeneous or slug flow regimes where particles are distributed along the length of the pipe or packed together as a large cluster respectively. In addition, due to the effects of gravitational forces which cause particles to settle towards the bottom wall of the horizontal pipe, the stratified and moving dunes flow regimes where particles are transported by traction along the pipe wall are observed at low gas velocities and solid concentrations. The solid concentration profile for stratified flow was unsymmetrical with higher concentration near the lower wall of the pipe while that for slug flow was similar to the flat profile seen for plug flow in vertical pneumatic conveying. The various flow regimes and their corresponding operating conditions have been represented in the form of phase diagrams.

Acknowledgements

This study has been supported by the National University of Singapore under the grant number R-279-000-095-112.

Literature Cited

1. Tashiro H, Peng X, Tomita Y. Numerical prediction of saltation velocity for gas-solid two-phase flow in a horizontal pipe. *Powder Technology.* 1997;91:141–146.
2. Huber N, Sommerfeld M. Modelling and numerical calculation of dilute-phase pneumatic conveying in pipe systems. *Powder Technology.* 1998;99:90–101.
3. Levy A. Two-fluid approach for plug flow simulations in horizontal pneumatic conveying. *Powder Technology.* 2000;112:263–272.
4. Cundall PA, Strack ODL. A discrete numerical model for granular assemblies. *Geotechnique.* 1979;29:47–65.
5. Tsuji Y, Tanaka T, Ishida T. Lagrangian numerical simulation of plug flow of cohesionless particles in a horizontal pipe. *Powder Technology.* 1992;71:239–250.
6. Tsuji Y, Kawaguchi T, Tanaka T. Discrete particle simulation of two-dimensional fluidized bed. *Powder Technology.* 1993;77:79–87.
7. Xu BH, Yu AB. Numerical simulation of the gas-solid flow in a fluidized bed by combining discrete particle method with computational fluid dynamics. *Chemical Engineering Science.* 1997;52:2785–2809.
8. Mikami T, Kamiya H, Horio M. Numerical simulation of cohesive powder behavior in a fluidized bed. *Chemical Engineering Science.* 1998;53:1927–1940.
9. Kaneko Y, Shiojima T, Horio M. DEM simulation of fluidized beds for gas-phase olefin polymerization. *Chemical Engineering Science.* 1999;54:5809–5821.
10. Rhodes MJ, Wang XS, Nguyen M, Stewart P, Liffman K. Study of mixing in gas-fluidized beds using a DEM model. *Chemical Engineering Science.* 2001;56:2859–2866.
11. Li J, Mason DJ. A computational investigation of transient heat transfer in pneumatic transport of granular particles. *Powder Technology.* 2000;112:273–282.
12. Han T, Levy A, Kalman H. DEM simulation for attrition of salt during dilute-phase pneumatic conveying. *Powder Technology.* 2003;129:92–100.

13. Di Felice R. The voidage function for fluid-particle interaction systems. *International Journal of Multiphase Flow.* 1994;20:153–159.
14. Xu BH, Yu AB, Chew SJ, Zulli P. Numerical simulation of the gas-solid flow in a bed with lateral gas blasting. *Powder Technology.* 2000;109:13–26.
15. Rao SM, Zhu K, Wang CH, Sundaresan S. Electrical capacitance tomography measurements on the pneumatic conveying of solids. *Industrial and Engineering Chemistry Research.* 2001;40:4216–4226.
16. Zhu K, Rao SM, Wang CH, Sundaresan S. Electrical capacitance tomography measurements on vertical and inclined pneumatic conveying of granular solids. *Chemical Engineering Science.* 2003;58:4225–4245.