MODELING OF HOPPER DISCHARGE AND PNEUMATIC CONVEYING USING THE DISCRETE ELEMENT MODELING APPROACH

S. Gupta\textsuperscript{ε}, V. Singh\textsuperscript{Ψ} and Simon Lo\textsuperscript{Ψ}

\textsuperscript{ε}CD-ADAPCO, Trident House, Basil Hill Road, Didcot, OX11 7HJ, U.K.
\textsuperscript{Ψ}Summer Student (2009), CD-adapco India, B.Tech 3\textsuperscript{rd} year, Chemical Engg., I.I.T Guwahati, India.

ABSTRACT

Solid handling and processing are widely carried out in process and steel industries. For examples – charging of solid raw materials into a blast furnace; injection of solid coal particles into a blast furnace along with hot gas; conversion of solid coal into solid (porous) coke in a coke-making unit; transport of coal, pneumatic conveying of sintered pellets, etc. Modeling the flow of these solids employing continuum approach has proved to be a big challenge for the researchers. Proper insight into the mechanics that cause the motion of the granular material is still not achieved. Recently, a Lagrangian modeling technique called Discrete Element Modeling approach (DEM) has gained tremendous popularity in the research circle owing to the ever increasing power and speed of modern computers. In DEM the motion of each and every particle is tracked in time and space, based on the forces acting on it. We have found DEM is capable of modeling typical granular flow applications like hopper discharge flow and dilute-phase pneumatic conveying with reasonable accuracy. In pneumatic conveying it is found that pressure drop in the pipe increases with increasing particle-loading and fluid flow rates, and residence time of particle increases with decreasing fluid velocity.

KEY WORDS: Pneumatic conveying, hopper discharge, pressure drop, Meyer and Sellers correlation.

1. INTRODUCTION

In process industries, solid handling and processing are often involved at one stage or another. Solids may be present in form of raw materials, as in the iron and steel industry, or in form of finished products, as in the pharmaceutical industry. Modeling and scale up of equipments from laboratory scale to plant scale has been a challenge for researchers because of inadequacies in our knowledge on flow of granular materials. Continuum based theories have not been successful enough in this regard. Recent improvements in computing power has led to popularization of a Lagrangian modeling technique for modeling granular materials, called the Discrete Element Modeling approach (DEM). DEM can be used to study and obtain particle level details in the flow of the granular mass.

In an iron and steel industry, solid handling and transport are important operations to be carried out. Raw materials required to produce molten iron in a blast furnace are in solid form. Coal is another important ingredient in iron making, it has to be converted into coke in coking units and fed into the blast furnace. A better understanding of granular matter can lead to better designs in reactors, silo storage and conveying systems, improving the overall efficiency of the plant. Study of solid descent in an iron making blast furnace and formation of raceway in front of tuyeres can also be studied using DEM. Many researchers have used DEM to model raceways [1, 9] in blast furnace and thus improving our understanding of fluid flow profiles in the furnace. Solids, in form of pulverized coal are also injected into many blast furnaces.

DEM is a useful tool in modeling various granular and particulate flow phenomena. In this study, we examine the capability of DEM in modeling common granular flow operations like hopper flow discharge and dilute phase pneumatic conveying in pipes and channels. The DEM model available in the commercial CFD software STAR-CD was used in the study.
1.1 Hopper discharge is a classic granular flow problem. Discharge of coarse granular material has been modeled using DEM by many researchers [3, 6]. Some researchers have extended their discharge studies to predict granular segregation during discharge [10]. In the current study, we try and model the discharge rate of solids from a flat bottom hopper with a slit shaped opening and compare the results with the Meyers and Sellers correlation. No effects of interstitial air are considered as we would be dealing with coarse particles only. A stable discharge from hopper leads to a smooth operation in the plant. Such examples of discharge can be found in an iron making unit during blast furnace charge delivery, filling of coking units with coal, in solid storage and transport systems.

1.2 Pressure drop in gas-solid two phase flow has in the past received attention from numerous researchers. Pneumatic conveying in pipes and channels is one such example. A host of experimental data and correlation are available in literature for prediction of the pressure drop [4, 5, 7]. Presence of particles in a fluid stream leads to a higher pressure drop due to momentum exchange between the particles and the fluid. The pressure drop in a pipe, which determines the process energy requirement, is dependent on factors like particle properties size and shape and also the flow and fluid properties.

2. THEORY

The DEM model in STAR-CD is based on the soft sphere modeling approach [2]. The linear momentum balance on a particle may be written as:

\[ m_i \frac{d\vec{v}_i}{dt} = m_i \vec{g} + \sum_{j=1}^{k} \vec{F}_{Wall-Particle} + \vec{F}_{Particle-Particle} + \vec{F}_{FluidDrag} \]  

[1]

The inter-particle and wall-particle interactions are modeled using a simple spring dashpot model. More information about the model can be found in [8].

Angular momentum of the particle may be written as:

\[ I_i \frac{d\vec{\omega}_i}{dt} = \sum_{j=1}^{k} \tau_{ij}, \quad k = \text{number of near-neighbors of } 'i' \]  

[2]

Fluid drag on the particle is given by the expression:

\[ F_{FluidDrag} = \frac{1}{2} C_D \rho_f A_d |v_f - v_i| (v_f - v_i) \]  

[3]

where the drag coefficient is given by:

\[ C_D = \frac{24}{Re} \left(1 + A \times Re^b\right) + \frac{C}{1 + \frac{D}{Re}} \]  

[4]

For spherical particles, values of constant \(A, B, C, D\) are 0.15, 0.687, 0, 0 respectively. Fluid flow is modeled using the standard Navier-Stokes equations and the k-\(\varepsilon\) model for turbulent flow.

3. RESULTS AND DISCUSSION

As mentioned earlier, in this paper we study two systems using DEM – hopper (DEM in absence of fluid flow) and horizontal pneumatic conveying (DEM coupled to fluid flow).
3.1 Hopper discharge study

The following values have been used in the DEM simulations for the particles, unless otherwise specified:

<table>
<thead>
<tr>
<th>Particle property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coefficient of restitution (particle-particle and wall-particle)</td>
<td>0.8</td>
</tr>
<tr>
<td>Coefficient of friction (particle-particle and wall-particle), static / kinetic</td>
<td>0.3 / 0.3</td>
</tr>
<tr>
<td>Density</td>
<td>2500 kg.m(^{-3})</td>
</tr>
<tr>
<td>Diameter</td>
<td>3 mm</td>
</tr>
</tbody>
</table>

Table 1: Particle properties used in Hopper Simulations.

Figure 1 shows a typical flat bottom hopper (0.1 X 0.3 X 0.025 m) used in this study. Mono-sized particles of size 3 mm are fed into the hopper and allowed to settle under gravity. Once all the particles have settled and come to rest, the hopper outlet is opened. Discharge rate of particles with time is studied.

![Figure 1: Rectangular flat bottomed hopper used in discharge study.](image)

Figure 2 shows DEM simulation snapshots of hopper filling and discharge. Particles are fed into the hopper from the top; particles settle at the bottom of the hopper and after 3 seconds. The hopper outlet, as seen in Figure 1, is opened after 3 seconds and the simulation run for another 7 seconds of real time. Figure 3 shows the total particle mass discharge with time from a hopper having an orifice opening of 25 mm. The cumulative mass discharged increases linearly with time and then decreases. The decrease occurs when majority of the particles have been discharged from the hopper. When the hopper is almost empty, fewer particles leave through orifice in comparison with the steady discharge period. The slope of the line in Figure 3 gives the discharge rate of particles from the hopper.
Figure 2: Snapshots of filling and emptying of a flat bottom-rectangular hopper. Color shows particle residence time on a scale of 0 – 10 seconds.
Theoretical discharge rate may be calculated using correlation by Meyers & Sellers for flat bottom rectangular hoppers:

\[ W = 1.03\rho_B g^{1/2}(L - kd_p)(B_o - k_o d_p)^{1/2} \]  

[5]

where \( \rho_B \) is the bulk density, the other symbols are defined in the nomenclature section. The bulk density is based on the solid fraction of particles around the hopper orifice. In the CFD model one can define a group of cells around the hopper opening, by finding particles present in these cells one can evaluate the local porosity around the orifice. This is done at different times during the steady discharge period so that an average value of local solid fraction is calculated and used in evaluating the bulk density.
packing height. The match between simulations and theoretical results is reasonable. The discharge rate decreases as the orifice size decreases, since smaller area is available for flow. Thus, we may conclude that the DEM simulations are able to predict discharge rate at different orifice openings with reasonable accuracy.

3.2 Horizontal pneumatic conveying study

In section 3.1, we studied a DEM application that did not involve fluid flow. In this section, we study a pneumatic conveying problem to validate DEM in presence of fluid flow. The pipe has a length of 1 m and an internal diameter of 52 mm. Default value of particle properties are given in Table 2.

<table>
<thead>
<tr>
<th>Particle property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coefficient of restitution (particle-particle</td>
<td>0.8</td>
</tr>
<tr>
<td>and wall-particle)</td>
<td></td>
</tr>
<tr>
<td>Coefficient of friction (particle-particle and</td>
<td>0.3/0.3</td>
</tr>
<tr>
<td>wall-particle), static/kinetic</td>
<td></td>
</tr>
<tr>
<td>Density</td>
<td>1050 kg.m$^{-3}$</td>
</tr>
<tr>
<td>Diameter</td>
<td>2.385 mm</td>
</tr>
<tr>
<td>Fluid drag model</td>
<td>standard</td>
</tr>
</tbody>
</table>

Table 2: Particle properties used in Pneumatic Conveying Simulations.

Figure 5 shows pressure drop per unit length in the pipe, at a particle flow rate of 251, 743 and 1244 kg/hr. The computed results are compared with experimental results given in [5]. It can be seen that the pressure drop increases with increasing gas velocity as well as increasing solid loading, as expected. The match between the simulations and experimental results is also reasonable. It should be noted that particle properties were not available and hence they were fixed as outlined in Table 2. The model predictions are sensitive to particle initial condition and particle properties. Lowering the coefficient of restitution would imply more energy loss for the particle and hence more momentum exchange with the fluid, thus increasing the overall pressure drop in the pipe. Same can be said in view of increasing particle-particle or wall-particle friction too.

![Figure 5: Pressure drop in dilute-phase pneumatic conveying at different fluid velocities and particle loadings.](image.png)

The residence time of particles in the pipe can also be studied using DEM. Figures 6a-c show residence time of particles exiting the pipe after 4 seconds in the 251 kg/hr particle flow rate case. It
can be seen that the residence time of particles decreases as the fluid flow velocity increases. At all three fluid flow-rates, there can be observed a wide range of particle residence time at the outlet. In the 32 m/s fluid velocity case, the particles are uniformly suspended in fluid at the pipe outlet and the residence time varies from 0.06 to 0.2 seconds. Particles with higher residence time would have undergone higher number of collisions in the pipe (particle-particle collisions and wall-particle collisions). In the 20 m/s fluid velocity case too, the particles are uniformly suspended at the pipe outlet no settling due to gravity can be observed. The residence time varies from 0.11 to 0.39 seconds. Some signs of settling can be seen in the case of fluid velocity 12 m/s. The particles concentration in the lower part of the pipe is higher than the upper part. The residence time of particle varies from 0.2 to 0.6 seconds. Particles with higher residence time are found in the lower portion of the pipe (dense region). This is expected as particles there have a higher probability of undergoing collisions than in lean flow regions.

Figure 6a: Fluid inlet velocity of 12 m/s
Figure 6b: Fluid inlet velocity of 20 m/s
Figure 6c: Fluid inlet velocity of 32 m/s

Figure 6: Particle residence time at pipe outlet after 4 seconds of real time, 251 kg/hr particle loading.

4. CONCLUSIONS

DEM is able to predict the hopper discharge rate and pressure drop during dilute pneumatic conveying, with reasonable accuracy. Pressure drop in pneumatic conveying increases with increasing fluid flow-rate as well as increasing particle loading. Particles with a range of residence time or life time exist in the pipe. The residence time of particle increases with decreasing fluid flow-rate. It is found that particles near pipe wall tend to have larger residence time, which could be due to the higher probability of wall-particle collisions there.

5. NOMENCLATURE

\[ A_p \] Projected area of particle [m²].
\( A, B, C, D \) Constants in expression for \( C_D \). [-]

\( B_o \) Width of orifice opening [m].

\( C_D \) Drag Coefficient [-].

\( d_p \) Particle diameter [m].

\( \vec{F}_{\text{Wall-Particle}} \) Wall-particle interaction force [N].

\( \vec{F}_{\text{Particle-Particle}} \) Inter-particle interaction force [N].

\( \vec{F}_{\text{Fluid-Particle}} \) Fluid-particle interaction force [N].

\( g \) Acceleration due to gravity [ms^{-2}].

\( I_i \) Mass moment of inertia [kgm^{2}].

\( k_o \) constant = 1.5 for spherical particles [-].

\( L \) Length of orifice opening [m].

\( m_i \) Mass of particle 'i' [kg].

\( \text{Re} \) Particle Reynolds number [-].

\( t \) Time [s].

\( \vec{v}_i \) Velocity vector of particle 'i' [m/s].

\( \vec{v}_f \) Velocity vector of fluid [m/s].

\( W \) Discharge rate [kgs^{-1}].

\( \rho_B \) Bulk density [kgm^{-3}].

\( \rho_f \) Fluid density [kgm^{-3}].

\( \omega_i \) Angular velocity vector of particle [rad/s].

\( \tau_{ij} \) Net torque acting on particle 'i' due to neighbor 'j' [Nm].

6. REFERENCES