Refereed Proceedings

The 12th International Conference on Fluidization - New Horizons in Fluidization Engineering

Engineering Conferences International *Year* 2007

Discrete Particle Simulation of the Gas-Solid Flow in a Circulating Fluidized Bed

K. W. Chu* B. Wang† Aibing Yu‡

*The University of New South Wales
†The University of New South Wales
‡The University of New South Wales, A.yu@unsw.edu.au

DISCRETE PARTICLE SIMULATION OF THE GAS-SOLID FLOW IN A CIRCULATING FLUIDIZED BED

K. W. Chu, B. Wang and A. B. Yu
Centre for Simulation and Modeling of Particulate Systems
School of Materials Science and Engineering
The University of New South Wales
Sydney, NSW 2052
Australia

ABSTRACT

This paper presents a numerical study of the gas-solid flow in a three-dimensional Circulating Fluidized Bed (CFB) by means of Combined Continuum and Discrete Method (CCDM) in which the motion of discrete particles is described by Discrete Particle Method (DPM) on the basis of Newton's laws of motion applied to individual particles and the flow of continuum fluid by the traditional Computational Fluid Dynamics (CFD) based on the local averaged Navier-Stokes equations. The simulation is achieved by incorporating DPM codes into the commercial CFD software package Fluent. It is shown that the discrete particle simulation can capture the key flow features in CFB such as core-annulus structure, axial solid segregation and S-shaped axial solid concentration. The numerical results also show the effect of the pulsation arising from the expansion of the fluidized bed on the performance of the cyclone separator. The gas-solid, particle-wall and particle-particle interactions are analysed to understand the underlying mechanisms of CFB systems.

INTRODUCTION

Circulating fluidized beds (CFBs) are widely used in petroleum, power and nuclear industries. The mechanisms governing the gas-solid flow in CFBs are very complicated and not well understood although extensive experimental and numerical studies have been carried out in the past (see, for example, (1)).

Numerical methods have been widely used to study the fluid-solid flow in fluidization in recent years. The popular mathematical models proposed thus far can be grouped into two categories: the continuum-continuum approach at a macroscopic level represented by the so called two fluid model (TFM), and the continuum-discrete approach at a microscopic level mainly represented by the so-called combined continuum and discrete model (CCDM) which is sometime refereed to as computational fluid dynamics and discrete particle method (CFD-DPM) (2). In CCDM, the motion of individual particles is obtained by solving Newton's equations of motion while the flow of continuum gas is determined by the computational fluid dynamics on a computational cell scale. CCDM has been used by numerous investigators to study the gas-solid flow since the work of Tsuji et al. ((3), as briefly summarized by Feng and Yu (4). However, to date, such studies are largely limited to simple flow conditions and/or in two dimensions. This is particularly true considering the modeling of the gas-solid flow in a CFB which consists of a fluidized bed, a cyclone separator, a solid collector and a return leg. Most of the previous numerical studies only considered fluidized beds. In order to overcome this problem, e.g. to account for the pulsation of solid phase from the return leg, some investigators started simulating the whole recirculation loop, including, for example, the continuum modeling by Chiesa et al. (5) and the discrete approach by Ibsen et al. (6). However, 2D model and simplified cyclone separator were used, which can not truly represent the reality.

In this work, the CCDM model in our centre is extended from 2D to 3D and from simple geometry to complex geometry to overcome the above modeling efficiency. The simulation tool is achieved by coupling the DPM codes with commercial CFD software package Fluent. The approach is validated by qualitatively comparing the numerical results against experimental ones, and the results are used elucidate the mechanisms governing the gas-solid flow in a CFB in terms of particle-fluid, particle-particle and particle-wall interactions.

MATHEMATICAL MODEL

The CCDM model employed for this work is in principle the same as that used in our previous studies of the gas-solid flow (2, 7, 8). The governing equations for solid phase are given by:

$$m_i \frac{\mathrm{d}\mathbf{V}_i}{\mathrm{d}t} = m_i \mathbf{g} + \sum_{j=1}^{k_i} \left(\mathbf{F}_{cn,ij} + \mathbf{F}_{dn,ij} + \mathbf{F}_{ct,ij} + \mathbf{F}_{dt,ij} \right) + \mathbf{F}_{d,i} \tag{1}$$

and

$$I_i \frac{\mathrm{d}\boldsymbol{\omega}_i}{\mathrm{d}t} = \sum_{j=1}^{k_i} \left(\boldsymbol{T}_{ij} + \boldsymbol{M}_{ij} \right) \tag{2}$$

where m_i, I_i, $\mathbf{V}_i$ and $\boldsymbol{\omega}_i$ are, respectively, the mass, moment of inertia, translational and rotational velocities of particle i. The forces involved are the gravitational force, $m_i\mathbf{g}$, inter-particle forces between particles i and j which include the contact forces $\mathbf{F}_{cn,ij}$ and $\mathbf{F}_{ct,ij}$, and viscous damping forces $\mathbf{F}_{dn,ij}$ and $\mathbf{F}_{dt,ij}$, and the fluid-solid viscous drag force, $\mathbf{F}_{d,i}$. Torques, $\mathbf{T}_{ij}$, are generated by the tangential forces and cause particle i to rotate, because the inter-particle forces act at the contact point between particles i and j and not at the particle centre. $\mathbf{M}_{ij}$ are the rolling friction torques that oppose to the rotation of the ith particle (9). Equations to determine these forces and torques can be found elsewhere (7, 8).

The governing equations of gas phase are the same as those used in TFM, given by

$$\frac{\partial(\varepsilon)}{\partial t}+\nabla\cdot(\varepsilon\boldsymbol{u})=0 \tag{3}$$

and

$$\frac{\partial(\rho_f\varepsilon\mathbf{u})}{\partial t}+\nabla\cdot(\rho_f\varepsilon\mathbf{uu})=-\nabla P-\frac{\sum_{i=1}^{k_c}\mathbf{F}_{f,i}}{\Delta V_c}+\nabla(\varepsilon\boldsymbol{\tau})+\rho_f\varepsilon\mathbf{g} \tag{4}$$

where ε, $\mathbf{u}$ and t are, respectively, porosity, fluid velocity and time; ρ_f, P and $\boldsymbol{\tau}$ are the fluid density, pressure, viscous stress tensor. $\mathbf{g}$ is the gravity acceleration. $\mathbf{F}_{f,i}$ is the force of particle j on fluid phase. ΔV_c and k_c are, respectively, the volume of a computational cell volume and the number of particles in the cell.

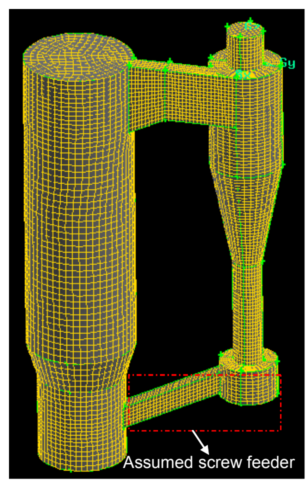


Fig. 1 Geometry and mesh representation of the simulated CFB

SIMULATION CONDITIONS

As shown in Fig. 1, a whole loop of CFB is simulated. 20,000 spherical particles with three diameters 0.0005, 0.000375 and 0.00025 m are initially packed at the bottom of the bed. The time step is 10^{-6} s for both solid and gas phases. Air is introduced from the bottom of the bed uniformly, with a superficial velocity at 5 m/s. For simplicity, a screw feeder is assumed in the return leg region by "dragging" the particles collected by the cyclone into the bed at a given speed 0.3 m/s. The particles (and wall as well) have the following properties: density 2500 kg/m^3, Young's modulus 1×10^7 N/m^2, sliding and rolling friction coefficients 0.3 and 0.005 respectively, and damping coefficient 0.3.

The 12th International Conference on Fluidization - New Horizons in Fluidization Engineering, Art. 90 [2007]

Non-slip boundary condition is applied to gas phase.

To take the advantages of the CFD development, we have extended our CCDM code with Fluent as a platform, achieved by incorporating a DPM code into Fluent through its User Defined Functions (UDF). The computational domain for particle and fluid phases is same, with the boundary meshes automatically generated in Fluent for a considered system. This approach allows particle-fluid flow under complicated conditions to be handled readily. The coupling of DEM and CFD at different time and length scales is achieved using the scheme same as that in our previous studies (7, 8).

RESULTS AND DISCUSSION

Fig. 2 shows the snapshots of the simulated CFB. Once gas is introduced, the bed begins to expand, with some particles hitting the top wall and falling down. Some particles close to the cyclone are dragged by gas into the cyclone. At the initial stage, the flow of particles into the cyclone is high, leading to particle accumulation at the apex of the cyclone. Some particles may exit the system from vortex finder due to strong particle-particle interaction. After t=0.6s, the number of particles flowing into the cyclone begins to decrease. Finally the in and out flows balance to give a macroscopically steady state flow. It can also be seen from Fig. 2 that there is obviously axial solid segregation characterized by the fact that large particles are mainly in the bottom part of the fluidized bed and small particles in the top part of the bed. Such results have been well documented in the literature (8).

Core-annulus flow structure in a CFB has been extensively reported in the literature and is characterized by the facts that solid concentration is higher near the wall than in the center, particles always move upward in the centre but can be either upward or downward near the wall, and gas velocity is high in the centre and low near the wall (10). These phenomena can be captured in the present simulation as shown in Figs. 3 and 4. From Fig. 3(a) it can be seen that there are much more particles near the wall than in the centre, especially near the wall close to the cyclone side. Figs. 3(a) and (b) also indicate that particles flow upward in the centre and almost all of the particles near the wall flow downward.

Following the previous studies (2, 4, 7, 8), the gas-particle and particle-particle interactions have been analyzed for the present CFB system. Fig. 4 shows the spatial distributions of gas flow and gas-particle interactions. It can be seen from Figs. 4(a) and (b) that gas velocity in the centre is much higher than near the wall. Furthermore, there is a region where gas flows downward. That gas may flow downward near the wall was suspected by some researchers but it is difficult to measure because of the presence of particles near wall (11). Figs. 4(b) and (d) on the other hand suggest that gas velocity is low or downward in a region where the gas-particle interaction force is high. This is because the strong action of solid phase on gas phase will detour the flow of gas and gas intends to flow through regions with low resistance.

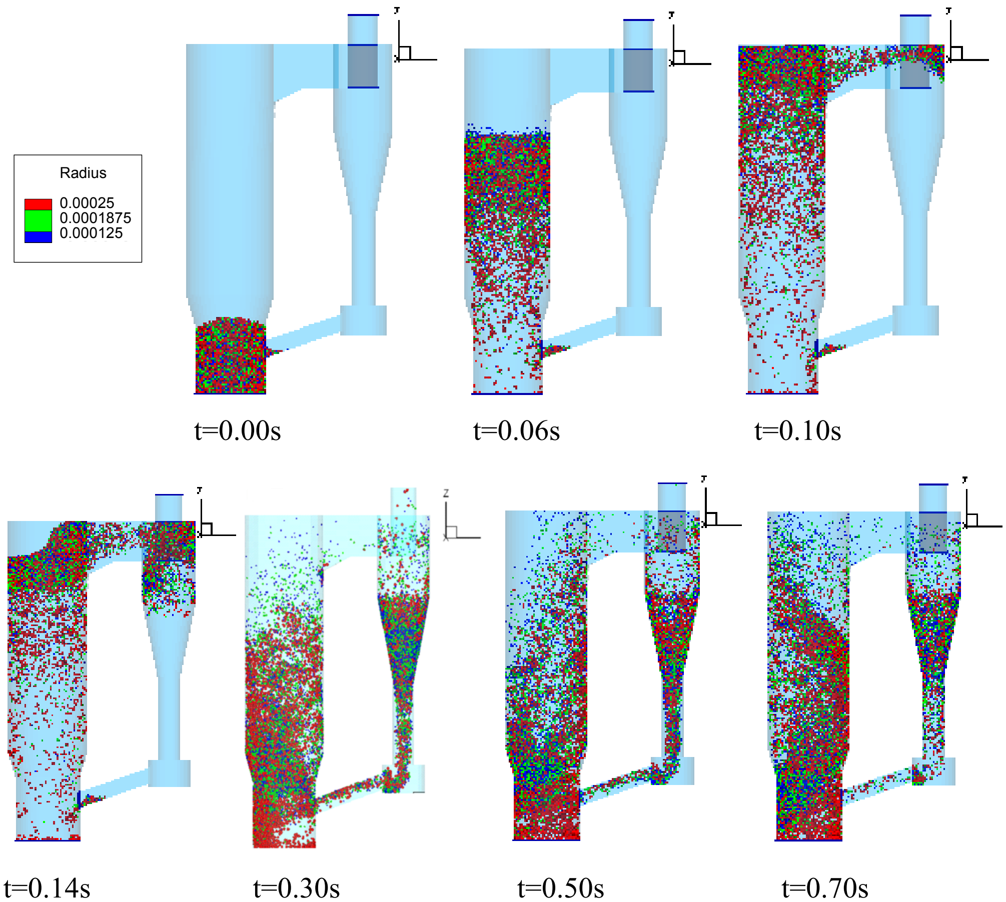
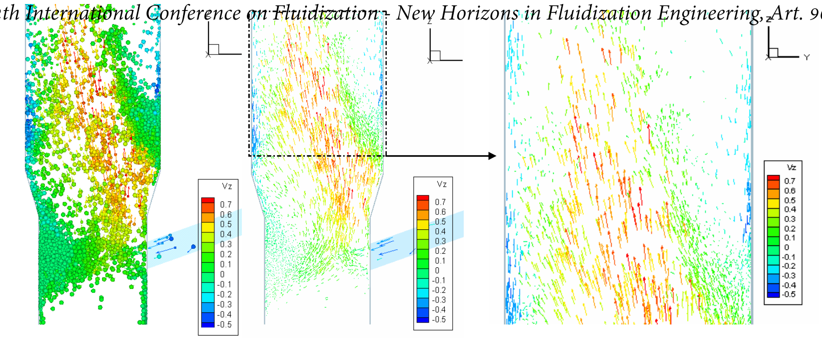


Fig. 2. Snapshots of simulated CFB (particles are coloured by size (m)).

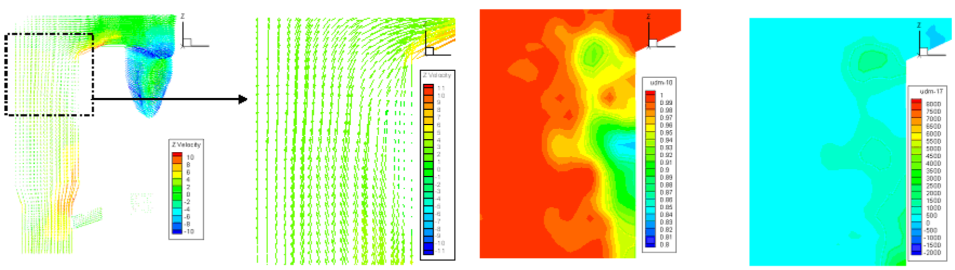
Comparison of Figs. 4(c) and (d) indicates that the distribution of porosity largely agrees with that of gas-particle interaction force although not always exactly. This is because the magnitude of gas-particle interaction force depends on not only the local porosity but also on the relative velocity between gas and particles. The effect of solid phase on gas phase can be described as follows. In pure fluid flow, the velocity near the wall is smaller than that in the centre because of fluid-wall friction (non-slip conditions). When particles are present in the bed, particles near the wall often move upward with slower velocities than those in the centre, causing particles to accumulate near the wall and generating a large resistance force to fluid flow. Consequently, fluid tends to flow through the bed centre. The more fluid flows through the centre, the more particles accumulate near the wall.

The 12th International Conference on Fluidization - New Horizons in Fluidization Engineering, Art. 90 [2007]



(a) (b) (c)

Fig. 3. Core-annulus flow structure in the CFB, colored by particle velocity (m/s) in z-direction: (a), particle position and velocity; (b) and (c), particle velocities at different scales.



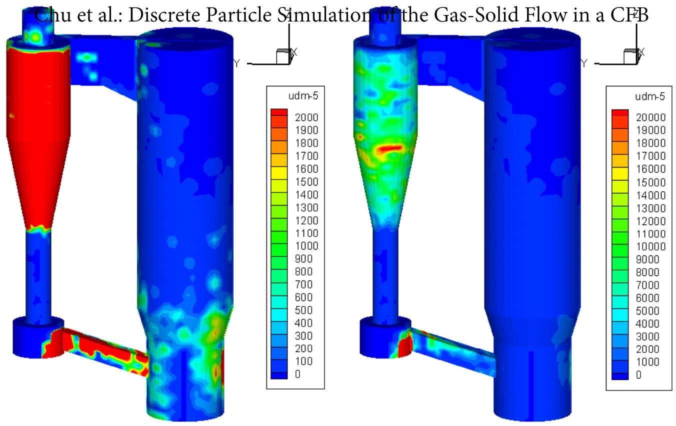
(a) (b) (c) (d)

Fig. 4. Gas-particle interaction at t= 0.9s: (a) and (b), gas velocities; (c), porosity; (d), gas-particle interaction force per unit volume.

Particle-wall interaction forces are important in the evaluation of wall erosion. In this work, Time Averaged Collision Intensity (TACI) is used to quantify wall erosion and defined as:

$$TACI = \frac{\sum_{t=0}^{t=T} \sum_{i=1}^{k_c} \mathbf{F}_{c,i}}{S_c \times T \times m_a g} \qquad (5)$$

where S_c is the surface area of a wall sample, T is the simulation time or sampling time, and $\mathbf{F}_{c,i}$ is the instantaneous contact force between particle i and wall. m_a is the average mass of all of the particles in the system. Fig. 5 shows the spatial distribution of TACI. It indicates that the most intensive interactions are on the bottom wall of the fluidized bed, cyclone apex wall and return leg. Accordingly, wall erosion may mainly occur to these regions in CFBs.



(a) (b)

Fig. 5. Time-averaged particle-wall interaction force over 1 second (the units are N/m^{2}s at different magnitude: (a), from 0 to 2000; and (b), from 0 to 20000.

There is little experimental information about inter-particle collision in a CFB, although such information is also useful to understanding the fundamentals governing the gas-solid flow. This difficulty can be readily overcome by the CCDM simulation. As an example, Fig. 5 shows the distribution of the particle-particle interaction force for the CFB considered. It appears that the most intensive particle-particle interactions are in regions with high solid concentration, e.g. at the cyclone apex. Further analysis is under way to understand its link to the gas-solid flow.

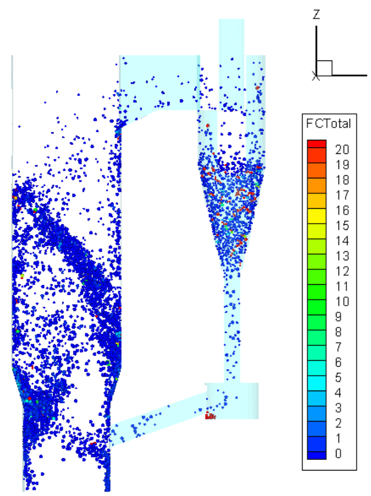


Fig. 6. A snapshot showing the spatial distribution of particle-particle interaction forces (the unit is the summed contacting forces dividing by the gravity of the heaviest particle in the system).

CONCLUSIONS

The CCDM approach has been extended to study the gas-solid flow in a whole loop of CFB system in three dimensions. It is shown that the model can satisfactorily capture the key features in a CFB such as the core-annulus flow structure, axial solid segregation and S-shaped solid axial concentration. The analysis of the gas-particle, particle-wall and particle-particle interaction forces leads to the following conclusions:

- The initial bed expansion can lead to the accumulation of particles at the apex of the cyclone and cause some particles to escape from the system.
- Gas-particle interaction force is strong near the wall of the fluidized bed, which may cause downward gas flow near the wall.

- Intensive particle-wall interaction forces mainly occur at the cone and cylinder regions of the cyclone separator, the bottom of the fluidized bed and the return leg in the feeder.
- Intensive particle-particle interaction forces occur at high solid concentration region, near wall region, and the apex region of the cyclone separator.

ACKNOWLEDGEMENT

The authors are grateful to Australian Research Council for the financial support of this work.

REFERENCES

1. Oka, S. N.; Anthony, E. J., *Fluidized Bed Combustion*. Marcel Dekker Inc.: New York and Basel, 2003; p 590.
2. Yu, A. B.; Xu, B. H., Particle-scale modelling of gas-solid flow in fluidisation. *Journal of Chemical Technology and Biotechnology* **2003,** 78, (2-3), 111-121.
3. Tsuji, Y.; Kawaguchi, T.; Tanaka, T., Discrete Particle Simulation Of 2-Dimensional Fluidized-Bed. *Powder Technology* **1993,** 77, (1), 79-87.
4. Feng, Y. Q.; Yu, A. B., Assessment of model formulations in the discrete particle simulation of gas-solid flow. *Industrial & Engineering Chemistry Research* **2004,** 43, (26), 8378-8390.
5. Chiesa, M.; Mathiesen, V.; Melheim, J. A.; Halvorsen, B., Numerical simulation of particulate flow by the Eulerian-Lagrangian and the Eulerian-Eulerian approach with application to a fluidized bed. *Computers & Chemical Engineering* **2005,** 29, (2), 291-304.
6. Ibsen, C. H.; Helland, E.; Hjertager, B. H.; Solberg, T.; Tadrist, L.; Occelli, R., Comparison of multifluid and discrete particle modelling in numerical predictions of gas particle flow in circulating fluidised beds. *Powder Technology* **2004,** 149, (1), 29.
7. Xu, B. H.; Yu, A. B., 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, (16), 2785-2809.
8. Feng, Y. Q.; Xu, B. H.; Zhang, S. J.; Yu, A. B.; Zulli, P., Discrete particle simulation of gas fluidization of particle mixtures. *AIChE Journal* **2004,** 50, (8), 1713-1728.
9. Zhou, Y. C.; Wright, B. D.; Yang, R. Y.; Xu, B. H.; Yu, A. B., Rolling friction in the dynamic simulation of sandpile formation. *Physica A* **1999,** 269, (2-4), 536-553.
10. Rhodes, M. J.; Sollaart, M.; Wang, X. S., Flow structure in a fast fluid bed. *Powder Technology* **1998,** 99, (2), 194-200.
11. Moran, J. C.; Glicksman, L. R., Mean and fluctuating gas phase velocities inside a circulating fluidized bed. *Chemical Engineering Science* **2003,** 58, (9), 1867-1878.