# Gas-solid turbulent flow in a circulating fluidized beds riser: numerical study of binary particle mixtures

# GAS-SOLID TURBULENT FLOW IN A CIRCULATING FLUIDIZED BED RISER; NUMERICAL STUDY OF BINARY PARTICLE MIXTURES

**Y. He[1,2], N.G. Deen[1,*], M. van Sint Annaland[1], and J.A.M. Kuipers[1]**

*1, Faculty of Science and Technology, University of Twente, The Netherlands*
*2, Department of Power Engineering, Harbin Institute of Technology, China*
**Corresponding author: E-mail: N.G.Deen@utwente.nl*

**Abstract** - A numerical simulation was performed on a turbulent gas-particle multi-phase flow in a circulating fluidized bed riser based on a hard-sphere discrete particle model (DPM) for the particle phase and the Navier-Stokes equations for the gas phase. The sub-grid scale stresses (SGS) were modeled with the SGS model proposed by Vreman (2004). The model enables the calculation of an arbitrary particle size distribution. In this work, binary mixtures of particles with different diameters are used in the simulation. From the numerical results it is found that the particles display a radial and axial diameter distribution. Small particles have a higher vertical particle velocity than the large particles. With increasing superficial gas velocity, the vertical particle velocity is increased. The average particle velocity and concentration vary both in the radial and axial directions. Finally, the numerical results are compared with the experimental and numerical results of Mathiesen et al (2000).

## INTRODUCTION

Gas-solid fluidization processes in circulating fluidized beds are widely applied in industry such as coal combustion, catalytic cracking of petroleum and nuclear industries etc. Characterization of gas-particle two-phase flow in a circulating fluidized bed riser is important for process optimization. The particle size distribution has a significant influence on the gas-solid flow dynamics. Depending on the Tokes number the particles can either enhance or suppress the turbulent flow field. Because the gas-solid two-phase flow in a circulating fluidized bed riser is a typical turbulent flow, it is important to characterize these effects.

To handle the complicated turbulent flow in a riser, various theoretical approaches have been proposed in the past. There are various models for the simulation of gas-particle multiphase flow. These models can be subdivided in two main classes: Euler-Euler and Euler-Lagrange models. The former model is also known as the two-fluid model and is used in many studies, because it requires relatively low computing resources. In this model, the solid phases as well as the gas phase are treated as interpenetrating continuous phases. The presence of each phase is described by a volume fraction. The latter model also known as the discrete particle model (DPM) in which the dynamics of the gas phase is modeled in conjunction with a discrete description of each individual particle in the system. The interactions amongst the particles and between the particles and the gas phase are accounted for in the DPM. The particle dynamics are modeled with a hard-sphere model, soft-sphere model or a DSMC (direct simulation Monte Carlo) method. Compared with the discrete particle model, the two-fluid model cannot reveal the motion of individual particles. The DSMC method has large memory requirements, so it can only deal with reactors of very small size. The soft-sphere model requires small computational time steps to guarantee an accurate description of the collisions, which becomes very restrictive in systems with fast moving particles, such as risers. Hence, in this work the hard-sphere model was chosen as the method to simulate riser flow with large numbers of particles. Due to the continuous progress in computer development, DPM methods have become more and more popular in the simulation research work for fluidized bed.

Most studies of riser flow however are concerned with the Eulerian model to save computation time. This model is able to predict the formation of clusters and time-averaged solid concentration and flux distribution in circulating fluidized beds. In order to get a better understanding of cluster formation and behavior, a few of researchers used Lagrangian models for the simulation of the multiphase flow in fluidized beds (Tsuji, Tanaka & Yonemura, 1998; Helland, Occelli, & Tadrist, 2000, Wang et al., 2005).

Segregation of particles is an important phenomenon in many industries. Particles of different size have different effects on the flow behavior. To shed some light on these effects, most researchers started of by investigating binary mixtures. Jenkins and Mancini (1987) extended the kinetic theory for granular flow with corrections of the granular temperature for the individual phases. Gidaspow et al. (1996) and Manger (1996) extended the kinetic theory to binary solids mixtures applying a separate granular temperature for each particle phase. Applying the latter model, Mathiesen et al (1999, 2000) studied the particle dynamics of binary mixtures with various sizes in a riser. Lu & Gidaspow (2003) used a similar model to study binary granular mixtures. In the Lagrangian model, an arbitrary particle size or density distribution can be used, because the physical properties can be defined for each individual particle. Lagrangian models are more straightforward to apply for studying the flow with binary particle mixtures, since it is relatively easy to obtain the behavior of different particle phases and the distribution and segregation of particles. For this reason, this method is becoming more and more popular to simulate the flow behavior in bidisperse gas-solid fluidized beds.

In this work a hard-sphere discrete particle model is used to predict the turbulent flow in a circulating fluidized bed riser. The applied hard sphere model was first reported by Hoomans et al. (1996). In our simulations we applied the SGS model of Vreman (2004) to model the sub-grid scale stresses, as it implicitly accounts for the reduced turbulent viscosity near walls. Binary mixtures of particles with various diameters are introduced into the bed to investigate the particle behavior as a function of the particle size. Furthermore, the influence of particle size and size distribution on the flow pattern in a circulating fluidized bed riser will be discussed. Finally, the numerical results are compared with experimental and numerical data of Mathiesen et al. (2000).

## GOVERNING EQUATIONS

The gas flow is modeled by the volume-averaged Navier-Stokes equations:

$$\frac{\partial(\varepsilon_g \rho_g)}{\partial t} + \nabla \cdot (\varepsilon_g \rho_g \mathbf{u}_g) = 0 \tag{1}$$

$$\frac{\partial(\varepsilon_g \rho_g \mathbf{u}_g)}{\partial t} + \nabla \cdot (\varepsilon_g \rho_g \mathbf{u}_g \mathbf{u}_g) = -\varepsilon_g \nabla P - \mathbf{S}_p - \nabla \cdot \left(\varepsilon_g \boldsymbol{\tau}_g\right) + \varepsilon_g \rho_g \mathbf{g} \tag{2}$$

Here, $\varepsilon_g$ is the porosity, and $\rho_g$, $\mathbf{u}_g$, $\boldsymbol{\tau}_g$ and $P$ respectively are the density, velocity, viscous stress tensor and pressure of the gas phase, respectively. The source term $\mathbf{S}_p$ is defined as:

$$\mathbf{S}_p = \frac{1}{V} \int \sum_{a=0}^{N_{part}} \frac{\beta V_a}{1-\varepsilon_g} (\mathbf{u}_g - \mathbf{v}_a) \delta(\mathbf{r} - \mathbf{r}_a) dV \tag{3}$$

Here $V$ is the volume of the fluid cell, $V_a$ the volume of particle, $\mathbf{v}_a$ the particle velocity, and $N_{part}$ the number of particles. The distribution-function $\delta$ distributes the reaction force of the particles exerted on the gas phase to the velocity nodes on the (staggered) Eulerian grid. To calculate the interphase momentum exchange coefficient $\beta$ we employed the well-known Ergun (1952) equation for porosities lower than 0.8 and the Wen and Yu (1966) correlation for porosities higher than 0.8 (Gidaspow, 1994).

$$\beta = \begin{cases} 150 \dfrac{\mu_g \varepsilon_s^2}{\varepsilon_g^2 d_p^2} + 1.75 \dfrac{\rho_g \varepsilon_s}{\varepsilon_g d_p} \left| \mathbf{u}_g - \mathbf{v}_a \right| & \forall \varepsilon_g \le 0.8 \\ \dfrac{3 C_d \varepsilon_s \varepsilon_g \rho_g \left| \mathbf{u}_g - \mathbf{v}_a \right|}{4 d_p} \varepsilon_g^{-2.65} & \forall \varepsilon_g > 0.8 \end{cases} \tag{4}$$

with

$$\mathrm{Re}_p = \frac{\rho_g \left| \mathbf{u}_g - \mathbf{v}_a \right| d_p}{\mu_g} \tag{5}$$

$$C_d = \begin{cases} \dfrac{24}{\mathrm{Re}} \left( 1 + 1.15\, \mathrm{Re}^{0.687} \right) & \forall\, \mathrm{Re} < 1000 \\ 0.44 & \forall\, \mathrm{Re} \ge 1000 \end{cases} \tag{6}$$

where $\mathrm{Re}_p$, $d_p$ and $\mathbf{v}_a$ are respectively the particle Reynolds number, diameter and velocity. $\varepsilon_s$ and $C_d$ are the local solids volume fraction and the drag coefficient respectively. It is known that this drag relation has a discontinuity around the switching value of $\varepsilon_g = 0.8$. However, since the local instantaneous particle volume fraction is generally below $\varepsilon_s = 0.2$ this has no effect on the simulation results.

The eddy viscosity of the gas is calculated by the SGS model of Vreman (2004):

$$\nu_e = c\sqrt{\frac{B_\beta}{\alpha_{ij}\alpha_{ij}}} \tag{7}$$

with

$$\alpha_{ij} = \frac{\partial \mathbf{u}_{g,j}}{\partial x_j},\ \beta_{ij} = \Delta_m^2 \alpha_{mi}\alpha_{mj},\ B_\beta = \beta_{11}\beta_{22} - \beta_{12}^2 + \beta_{11}\beta_{33} - \beta_{13}^2 + \beta_{22}\beta_{33} - \beta_{23}^2 \tag{8}$$

Here $\nu_e$ is the eddy viscosity of gas phase. $c$ and $\Delta_m$ are a model constant and the local filter width, which will be explained later. The symbol $\alpha$ stands for the (3×3) matrix of derivatives of the filtered velocity $\mathbf{u}_g$. We define that if $\| \alpha_{ij}\alpha_{ij} \| = 0$, then $\nu_e = 0$.

The model constant $c$ is related to the Smagorinsky constant $C_s$ by $c \approx C_s^2$. In our simulation we let $C_s = 0.1$ (i.e. c = 0.025). The applied turbulence models can be implemented in the DPM straightforwardly, since it only requires the local filter width and the first-order derivatives of the velocity field. As of yet, the effect of the particles on the turbulence is not well understood and is therefore ignored in this work. The gas phase equations are solved numerically using with a finite differencing technique, in which a staggered grid was employed to ensure numerical stability. The applied numerical grid was selected in such way that all relevant structures are resolved, while ensuring that the particle volumes are significantly smaller than the volume of a grid cell.

The hard-sphere discrete particle model (DPM) used in this work was originally developed by Hoomans et al. (1996). In the DPM the particles are assumed to be rigid spheres moving in free flight. When collisions amongst particles occur, these are treated as binary, instantaneous, impulsive events.

The velocity of every individual particle can be calculated from Newton's second law, containing forces due to the pressure gradient, drag and gravitation:

$$\frac{m_a d^2\mathbf{r}_a}{dt^2} = \frac{V_a\beta}{1-\varepsilon}\left(\mathbf{u}_g - \mathbf{v}_a\right) - V_a\nabla P + m_a\mathbf{g} \tag{12}$$

$$I_a\mathbf{\Omega}_a = I_a\frac{d\boldsymbol{\omega}_a}{dt} = \mathbf{T}_a \tag{13}$$

Here, $m_a$ is the mass of the particle, $\mathbf{T}_a$ the torque, $I_a$ the moment of inertia, $\mathbf{\Omega}_a$ the rotational acceleration, and $\boldsymbol{\omega}_a$ the rotational velocity.

In this model, it is assumed that the interaction forces are impulsive and therefore all other finite forces are negligible during collision. The particle collision characteristics play an important role in the overall system behavior. For this reason realistic collision properties of the particles are supplied to the model, i.e. the coefficients of the normal and tangential restitution are respectively set to $e_n = 0.97$ and $\beta_0 = 0.33$, and the coefficient of friction is set to $\mu = 0.1$ for both collisions amongst particles and with the confining walls.

## INITIAL AND BOUNDARY CONDITIONS

A sketch of the fluidized bed riser used in this study is shown in Figure 1. The simulations are carried out only for the central part of the riser without considering the inlet and exit effects. Initially, the particles are distributed evenly across the flow domain to achieve a uniform initial concentration distribution. Each particle that crosses one of the top or bottom boundaries is inserted again at the opposite boundary while retaining all of its physical properties. The use of these boundary conditions implies that the total number of particles in the flow domain as well as the overall particle volume fraction remains constant. Gas is injected at a constant uniform flow rate at the bottom of the column and a pressure boundary condition is used for the gas phase at the top of the bed. No-slip conditions are used for the gas phase at the left and right walls, while free slip boundary conditions are applied at the front and back wall. The physical parameters are chosen in accordance with the work of Mathiesen et al. (2000), who carried out a combined experimental and numerical study of the flow in a riser. In our work, the channel height is less than that in the case of Mathiesen et al., so the number of the particles was reduced to match the overall particle volume fraction. The computed parameters for the simulations are listed in table 1. The average volume fraction for the small and large particles is equal. All simulations were run for 8 s and time averages were calculated during the last 5 s.

Table 1. Parameters used in the basic case.

| Parameter | Value | Unit |
|---|---|---|
| Particle diameter, $d_p$ | 120 & 185 | (μm) |
| Average particle concentration, $\varepsilon_s$ | 1.25 & 1.25 | % |
| Particle density, $\rho_p$ | 2400 | (kg/m$^3$) |
| Normal restitution coefficient, $e_n$ | 0.97 | (-) |
| Tangential restitution coefficient, $\beta_0$ | 0.33 | (-) |
| Friction coefficient, $\mu$ | 0.10 | (-) |
| CFD time step | $5.0\times10^{-5}$ | (s) |
| Particle time step, $\Delta t$ | $5.0\times10^{-5}$ | (s) |
| Channel length, $D$ | 0.032 | (m) |
| Channel width, $W$ | 0.0012 | (m) |
| Channel height, $H$ | 0.30 | (m) |
| CFD grid number, $Nx$ | 25 | (-) |
| CFD grid number, $Ny$ | 1 & 6 | (-) |
| CFD grid number, $Nz$ | 60 | (-) |
| Shear viscosity of gas, $\mu_g$ | $1.8\times10^{-5}$ | (Pa·s) |
| Gas temperature, $T$ | 313 | (K) |
| Pressure, $P$ | 1.2 | (Bar) |
| Velocity, $u_g$ | 1.0 & 1.2 & 2.0 | (m/s) |
| Particle terminal velocity | 0.92 & 1.42 | (m/s) |
| (120 μm &185 μm), $u_t$ | | |
| Number of particles, $N_p$ | 202,600 | (-) |

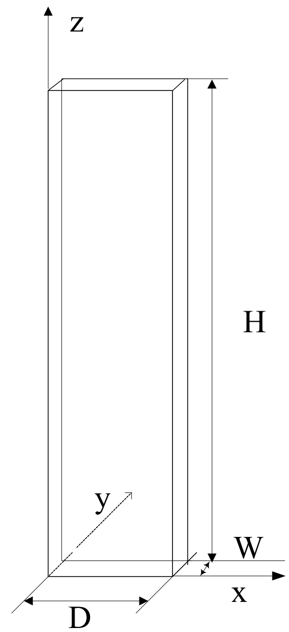
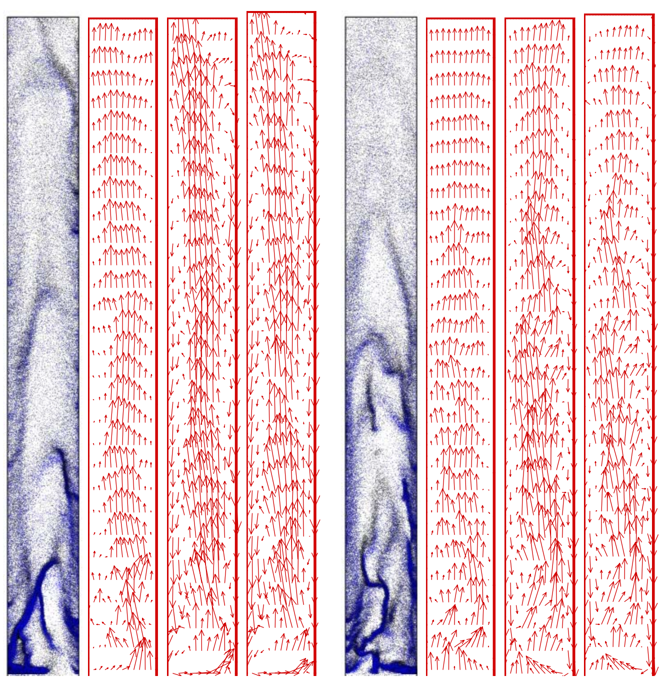



*Fig 1: Schematic representation of the fluidized bed riser.*

## SIMULATION RESULTS

Figure 2 shows a time series of the flow patterns and the velocities of the gas phase in the riser resulting from the simulations. The particles from horseshoe shaped clusters that move both in upward and downward directions. The clusters form, grow up, change their shape and finally break up. After breaking up, the particles are collected near the walls. The particles tend to concentrate near the bottom of the bed. The velocities of the gas phase are all positive, showing the largest values in the core of the bed. The gas velocity is considerably reduced in the vicinity of the clusters.



t = 1.0 s    t = 2.0 s

*Fig. 2: From left to right: snapshots of the particle distribution and velocity fields of the gas, small particle (d = 120 μm) and large particle (d = 185 μm) phases at two different time instances.*

Particles are moving upward in the bed center and flow downward close to the walls. From this figure, the status of clusters can be clearly observed. Particles that are inside a cluster flow with low velocities. A typical core-annulus flow pattern is observed in the snapshots. The flow patterns for the particles of the two size classes are nearly the same, while the velocity and volume fraction of particles differ.

Figure 3 shows the average vertical particle velocity at different heights. A typical core-annulus flow structure is observed, that is to say that upflow takes place in the centre of the riser, while sharp velocity gradients and particle downflow are found near the walls. The latter can be related to the passage of clusters, which are mostly found near the walls. The simulated results agree well with the experimental and numerical results of Mathiesen et al. (2000). Our simulation results are better than the computational result of Mathiesen et al. (2000) especially in the zone near the bottom of the bed when $z/H = 0.2$. It is seen that the DPM is better able to predict the small velocity differences between the particles than the multi-fluid model of Mathiesen et al. (2000). The asymmetry in the experimental data is probably related to inlet effects.

Figure 4 shows volume fraction profiles of particles at different heights in the riser. The concentration of particles is low in the bed center and high near the walls where clusters are mostly found. The particle volume fraction decreases with increase in bed height. The numerical result is compared with the experimental and numerical results of Mathiesen et al. (2000). All results show the same tendencies.

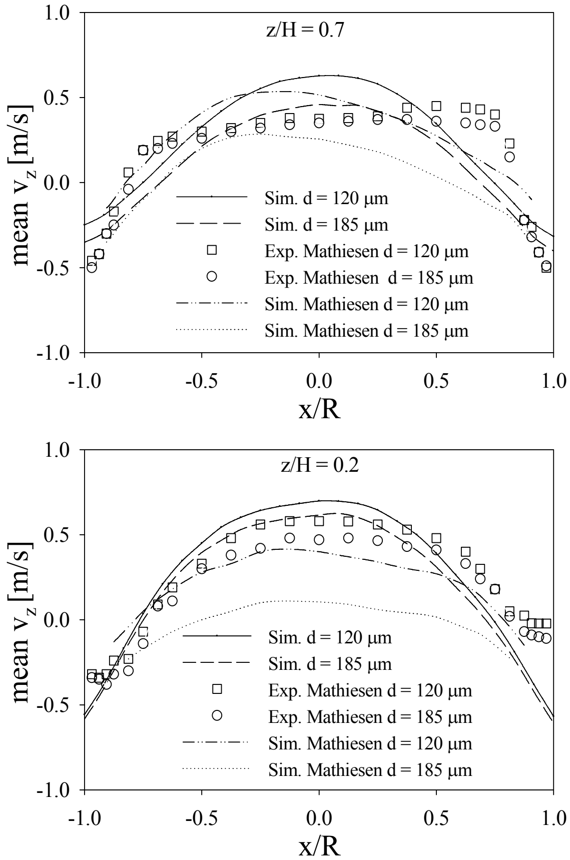



*Fig. 3: Particle vertical velocity profiles at different heights.*

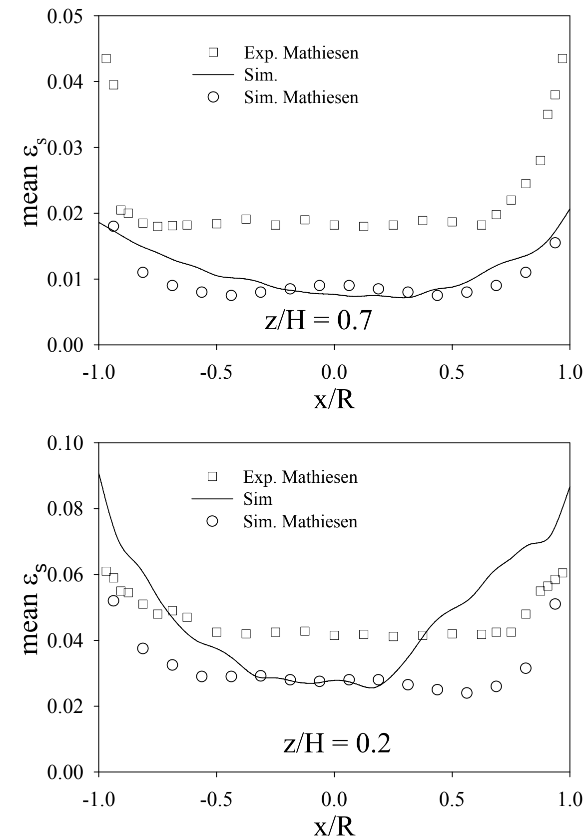



*Fig. 4: Mean volume fraction profiles of particles at different heights.*

## CONCLUSIONS

In this paper, we studied the turbulent gas-particle two-phase flow in a pseudo-2D circulating fluidized bed riser based on a full 3D DPM model. Because both the inter-particle collisions and the gas phase turbulence play important roles in these type of flows, we accounted for these effects in our simulations with the use of a large eddy simulation (LES) turbulence closure proposed by Vreman (2004) and four-way coupling. The following main conclusions were obtained from the work presented in this paper. The different particle phases show distinctively different behavior, that is to say that small particles experience a larger drag and thus obtain a higher vertical velocity and a more homogeneous axial distribution. Furthermore, the turbulent flow in the circulating fluidized bed riser is a typical nonlinear system with a chaotic character.

## ACKNOWLEDGEMENTS

This work was supported by the National Science Foundation through Grant No. 50376013 and NSFC-PetroChina Company Limited under the cooperative project No. 20490200.

## REFERENCES

Ergun, S., 1952. Fluid Flow through Packed Columns, Chemical Engineering Progress, 48, 89-94.

Gidaspow, D., Lu, H. and Manger, E., 1996. Kinetic theory of multiphase flow and fluidization: validation and extension to binaries. XIXth International Congress of Theoretical and Applied Mechanics, Japan.

Gidaspow, D., 1994. Multiphase Flow and Fluidization. Continuum and Kinetic Theory Description. Academic Press, New York.

Helland, E., Occelli, R. and Tadrist, L., 2000. Numerical study of cluster formation in a gas–particle circulating fluidized bed. Powder Technology, 110, 210-221.

Hoomans, B.P.B., Kuipers, J.A.M., Briels, W.J. and van Swaaij, W.P.M., 1996. Discrete particle simulation of bubble and slug formation in a two-dimensional gas-fluidised bed: A hard-sphere approach. Chemical Engineering Science, 51, 99-108.

Jenkins J.T. and Mancini F., 1989. Kinetic theory for binary mixtures of smooth, nearly elastic spheres. Physics of Fluids, 31, 2050–2057.

Lu, H. and Gidaspow, D., 2003. Hydrodynamics of binary fluidization in a riser: CFD simulation using two granular temperature. Chemical Engineering Science, 58, 3777-3792.

Manger, E., 1996. Modelling and simulation of gas/solid flow in curvilinear coordinates. Ph.D. thesis, Telemark Institute of Technology, Norway.

Mathiesen, V., Solberg, T., Arastoopour, H. and Hjertager, B.H., 1999. Experimental and computational study of multiphase gas/particle flow in a CFB riser. AIChE Journal, 45, 2503–2518.

Mathiesen, V., Solberg, T. and Hjertager, B.H., 2000. An experimental and computational study of multiphase flow behavior in a circulating fluidized bed. International Journal of Multiphase Flow, 26, 387-419.

Tsuji, Y., Tanaka, T. and Yonemura, S., 1998. Cluster patterns in circulating fluidized beds predicted by numerical simulation (discrete particle model versus two-fluid model). Powder Technology, 95, 254-264.

Vreman, A.W., 2004. An Eddy-viscosity Subgrid-scale Model for Turbulent Shear Flow: Algebraic Theory and Applications. Physics of Fluids, 16, 3670-3681.

Wang, S., Liu, H., Lu, H., Liu, W., Jiamin, D. and Li, W., 2005. Flow behavior of clusters in a riser simulated by direct simulation Monte Carlo method. Chemical Engineering Journal, 106, 197-211.

Wen, Y.C. and Yu, Y.H., 1966. Mechanics of Fluidization, Chemical Engineering Progress Symposium Series, 62, 100-111.