

*2007 ECI Conference on The 12th International Conference on  
Fluidization - New Horizons in Fluidization Engineering*

Vancouver, Canada

Editors: Franco Berruti, The University of Western Ontario, London,  
Canada

Xiaotao (Tony) Bi, The University of British Columbia, Vancouver, Canada

Todd Pugsley, University of Saskatchewan, Saskatoon, Saskatchewan,  
Canada

---

*Year 2007*

*Paper 15*

---

A Computational Study of the Distribution of  
Particles in a Lab-Scale CFB Boiler

Kim G. Hansen\*      Tron Solberg<sup>†</sup>

Bjorn H. Hjertager<sup>‡</sup>

\*Dong Energy Generation

<sup>†</sup>Aalborg University

<sup>‡</sup>Aalborg University

[http://services.bepress.com/eci/fluidization\\_xii/15](http://services.bepress.com/eci/fluidization_xii/15)

Copyright ©2007 by the authors.

# A Computational Study of the Distribution of Particles in a Lab-Scale CFB Boiler

## **Abstract**

When two-fluid modeling is used to predict riser flows there have been difficulties in predicting the solids hold up in risers represented by the correct pressure drop profile. A way of encountering this inherent problem in current Eulerian-Eulerian CFD modeling is to approximate the actual particle size distribution by using more particle phases instead of the current practice of using one mean diameter. For the lab-scale CFB investigated, CFD simulations show that a mal-distribution occurs in the CFB; the larger particles are retained in the riser, whereas the intermediate and small particles are distributed both in the return leg and the riser. Simulations using an altered particle size distribution, i.e. a larger amount of large particles, show significant improvements in the pressure profile in the bottom part of the riser.

## A COMPUTATIONAL STUDY OF THE DISTRIBUTION OF PARTICLES IN A LAB-SCALE CFB BOILER

Kim Granly Hansen<sup>1,2,\*</sup>, Tron Solberg<sup>1</sup> and Bjorn H. Hjertager<sup>1</sup>

<sup>1</sup>Chemical Engineering Laboratory

Aalborg University Esbjerg, DK-6700 Esbjerg, Denmark

<sup>2</sup>Dong Energy Generation, Project Development and Engineering

DK-7000 Fredericia, Denmark

\*T: +45 7622 2633; kigha@dongenergy.dk

### ABSTRACT

When two-fluid modeling is used to predict riser flows there have been difficulties in predicting the solids hold up in risers represented by the correct pressure drop profile. A way of encountering this inherent problem in current Eulerian-Eulerian CFD modeling is to approximate the actual particle size distribution by using more particle phases instead of the current practice of using one mean diameter. For the lab-scale CFB investigated, CFD simulations show that a mal-distribution occurs in the CFB; the larger particles are retained in the riser, whereas the intermediate and small particles are distributed both in the return leg and the riser. Simulations using an altered particle size distribution, i.e. a larger amount of large particles, show significant improvements in the pressure profile in the bottom part of the riser.

### INTRODUCTION

Circulating fluidized bed units are widely used in industry. The major processes are Fluid Catalytic Cracking (FCC) and Circulating Fluidized Bed Combustion (CFBC). The gas/particle flow inside a CFB unit is very complex. The distribution of the particles in the riser gives a core-annulus flow with particles flowing down along the walls and strands or clusters of particles moving along with dispersed particles in the centre of the riser. The particle distribution in the riser influences the chemical reactions and/or heat transfer properties. Computational fluid dynamics (CFD) analysis of multiphase systems has evolved over the last few years to become a strong tool in the design and development of industrial equipment. It is especially useful in complex geometries and under conditions that do not allow access with measuring equipment. Modelling of gas/particle flow has achieved some good results during the last 10 to 15 years, (e.g. Ding and Gidaspow (3); Samuelsberg and Hjertager (11); Mathiesen *et al.* (10) 2000; Ibsen *et al.* (8); Agrawal *et al.* (1); and Zhang and VanderHeyden (14)).

When two-fluid modeling is used to predict riser flows there have been difficulties in predicting the solids hold up in risers represented by the correct pressure drop profile. A way of encountering this inherent problem in current Eulerian-Eulerian CFD modeling is to approximate the actual particle size distribution by using more particle phases instead of the current practice of using one mean diameter. In the work of

Hansen (7) it was seen that when using three particle phases to model flow in a lab-scale CFB, the phase with the largest particle diameter does not circulate. This means that only the smaller and intermediate size particles are circulating, and the size distribution in the riser might be significantly different from the average size distribution for the whole system.

The present work concerns multi-fluid CFD modeling of a lab-scale CFB riser that was investigated by Ibsen *et al.* (8). The distribution of the different particle sizes in the riser is investigated by using three particle phases to represent the particle size distribution. The effect of using an altered particle size distribution is reported.

## THE CFB SYSTEM

The CFB is a 1/9-scale model of the Chalmers 12 MW CFB boiler (9), operated according to the simplified scaling laws proposed by Glicksman *et al.* (5). The dimensions of the riser of the scale model shown in Figure 1 is 0.19 m x 1.5 x 0.17 m corresponding to depth (x) x height (y) x width (z). A cyclone is used to separate the solids, which passes a particle seal designed as a bubbling bed, before being reintroduced in the lower part of the riser. The entrance of the cyclone is located at the rear of the riser, 1.2 m above the primary air distributor. Laser Doppler Anemometry (LDA) measurements were performed along with pressure drop measurements by Ibsen *et al.* (8). The particle inventory is a powder of spherical bronze particles with a mean volume length diameter of 45  $\mu\text{m}$ .

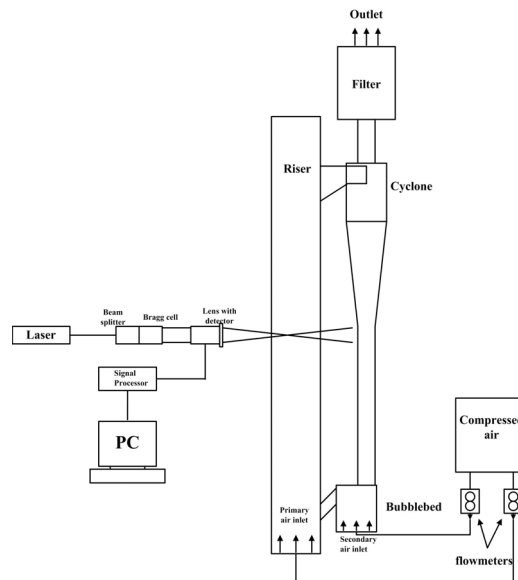


Figure 1. The CFB system studied by Ibsen *et al.* (8).

## NUMERICAL MODEL

The calculations are performed using the three-dimensional, finite-volume, multiphase Eulerian/Eulerian CFD code FLOTRACS-MP-3D, see also Mathiesen *et al.* (10), Ibsen *et al.* (8), and Hansen *et al.* (6). The turbulent motion of the particulate phase is modeled using the kinetic theory for granular flow, and the gas phase turbulence is modeled using a Sub-Grid-Scale model.

## Governing Equations

Using tensor notation the governing equations may be written as follows:

Continuity equation for phase  $k$ :

$$\frac{\partial}{\partial t}(\varepsilon_k \rho_k) + \frac{\partial}{\partial x_i}(\varepsilon_k \rho_k U_{i,k}) = 0 \quad (1)$$

where  $\varepsilon$ ,  $\rho$  and  $U$  are the volume fraction, density and velocity of phase  $k$ , respectively. Momentum equation for phase  $k$ :

$$\frac{\partial}{\partial t}(\varepsilon_k \rho_k U_{j,k}) + \frac{\partial}{\partial x_i}(\varepsilon_k \rho_k U_{i,k} U_{j,k}) = -\varepsilon_k \frac{\partial P}{\partial x_j} + \frac{\partial}{\partial x_i} \tau_{ij,k} + \varepsilon_k \rho_k g_j + \sum_{m=1, m \neq k}^M \beta_{km} (U_{j,m} - U_{j,k}) \quad (2)$$

Here  $P$ ,  $\tau_{ij}$ ,  $g$  and  $\beta$  are pressure, stress tensor, gravity and the inter-phase drag coefficient respectively. The gas phase stress tensor is given by:

$$\tau_{ij,g} = \mu_{eff,g} \left[ \left( \frac{\partial U_j}{\partial x_j} + \frac{\partial U_i}{\partial x_j} \right) - \frac{2}{3} \delta_{ij} \frac{\partial U_k}{\partial x_k} \right]_g \quad (3)$$

where  $\delta_{ij}$  is the Kroenecker delta. The effective viscosity,  $\mu_{eff,g}$ , is derived from a Smagorinsky sub-grid-scale (SGS) model, where the effective viscosity is a sum of a laminar and a turbulent part.

$$\mu_{eff,g} = \varepsilon_g (\mu_{lam,g} + \mu_{turb,g}) = \varepsilon_g \mu_{lam,g} + \varepsilon_g \rho_g (c_t \Delta)^2 \sqrt{S_{ij,g} : S_{ij,g}} \quad (4)$$

The SGS eddy coefficient,  $c_t$ , is set to 0.079 based on Deardoff (1). The length scale,  $\Delta$ , and the strain rate tensor of the resolved field,  $S_{ij,g}$ , are given by:

$$\Delta = (\Delta x \Delta y \Delta z)^{1/3} \quad \text{and} \quad S_{ij,g} = \frac{1}{2} \left[ \frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j} \right]_g \quad (5)$$

The solid phase stress tensor is given by:

$$\tau_{ij,s} = -P_s \delta_{ij} + \xi_s \delta_{ij} \frac{\partial U_{k,s}}{\partial x_k} + \mu_s \left[ \left( \frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j} \right) - \frac{2}{3} \delta_{ij} \frac{\partial U_k}{\partial x_k} \right]_s \quad (6)$$

The solids phase pressure,  $P_s$ , the bulk viscosity,  $\xi_s$ , and the shear viscosity,  $\mu_s$ , are derived from the kinetic theory of granular flow. The inter-phase drag coefficient is modeled after Gibilaro *et al* (3):

$$\beta_{sg} = \left( \frac{17.3}{\text{Re}_s} + 0.336 \right) \frac{\rho_g}{d_s} |\vec{u}_g - \vec{u}_s| \varepsilon_s \varepsilon_g^{-1.8} \quad (7)$$

where the particle Reynolds number is defined as:

$$\text{Re}_s = \frac{d_s \rho_g \varepsilon_g |\bar{u}_g - \bar{u}_s|}{\mu_{lam,g}} \quad (8)$$

A transport equation for the solids phase turbulent kinetic energy or granular temperature is solved:

$$\frac{3}{2} \left[ \frac{\partial}{\partial t} (\varepsilon_s \rho_s \Theta_s) + \frac{\partial}{\partial x_i} (\varepsilon_s \rho_s U_{i,s} \Theta_s) \right] = \frac{\partial}{\partial x_i} \left( \kappa_s \frac{\partial \Theta_s}{\partial x_i} \right) + \tau_{ij,s} : \frac{\partial U_{j,s}}{\partial x_i} - \gamma_s - 3\beta_{sg} \Theta_s \quad (9)$$

where  $\Theta_s$ ,  $\kappa_s$  and  $\gamma_s$  are the granular temperature, conductivity of the granular temperature and dissipation due to inelastic collisions respectively. The full model can be seen in Mathiesen et al. (10).

### Numerical Configuration

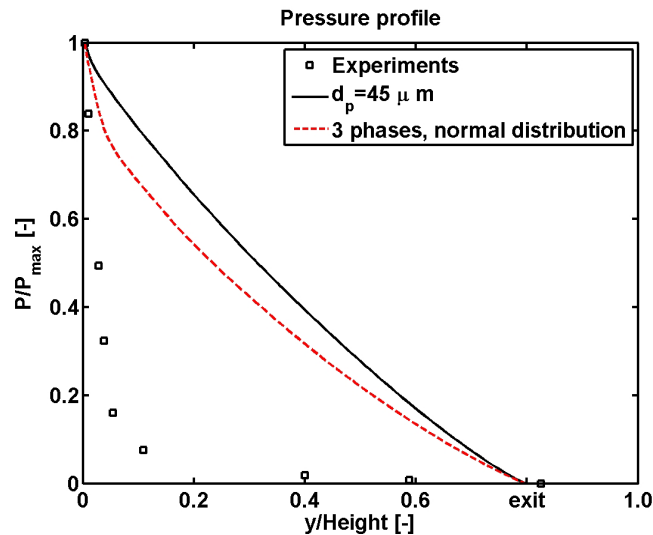
The physical domain that is simulated is equivalent to the one used by Ibsen *et al.* (8). A three-dimensional Cartesian geometry is chosen with dimensions 0.19x1.20x0.17 m<sup>3</sup>. The inlet is located at the bottom, and the outlet is located at the top, thereby neglecting the effects of the inlet and the exit, which are placed at the side on the actual riser. The grid used to simulate the Lab-scale CFB is uniform both in axial and radial direction with 81,000 cells (27x25x122). For all simulations a fixed inlet flux is used for the gas and particle phase(s). A simulation is run using one mean diameter of 45 μm (Case 1), which is equivalent to the case reported by Ibsen *et al.* (8). This case is compared to a case with three particle phases (Case 2). In order to see how a mal-distribution of particles in the system would influence the pressure drop in the riser, two cases with a larger amount of the bigger particles are run: One with double the amount of the largest phase (Case 3), and one with four times the amount of the largest phase (Case 4). Finally, In order to compare with the case of having only the largest particles present in the riser, a two-phase simulation is run where the particle phase only consists of particles having a diameter of 113 μm. The simulations performed are summarized in Table 1.

**Table 1. Overview of the simulations performed.**

Case No.	Number of internal cells	dp <sub>s</sub> [μm]
1	81,000 (27x25x122)	45 (100 %)
		20 (12 %)
2	81,000 (27x25x122)	63 (70 %)
		113 (18 %)
		20 (9.4 %)
3	81,000 (27x25x122)	63 (54.6 %)
		113 (36 %)
		20 (4.1 %)
4	81,000 (27x25x122)	63 (23,9 %)
		113 (72 %)
		113 (100 %)

## RESULTS AND DISCUSSION

Figure 2. Comparison of simulated and measured pressure profile. In the experiments a very large gradient is seen from  $y/\text{height} = 0$  to 0.3. This gradient arises from the bubbling or churning fluidizing zone in the bottom zone of the riser. The pressure drop is more constant in the upper part of the riser indicating a more dilute flow regime. The simulation using 1 particle diameter on the original grid (Case 1) gives an almost linear pressure drop profile. The case with a three particle phases (Case 2) shows an indication of the dense bottom zone seen in the experiments.



**Figure 2. Comparison of simulated and measured pressure profile.**

The indication of a sharp gradient found for this case can be explained by looking at Figure 3 and Figure 4. The distribution of the different particle phases can be seen in Figure 3. The smallest particle size is evenly distributed in the riser and the average volume fraction is around 0.005. The intermediate particle size shows a slight segregation in the riser whereas the largest particle phase is only found in the bottom of the riser from 0 to 0.1 m. The average volume fraction for the largest particle size above this height is  $10^{-6}$ . The segregation in size can also be seen in the computed mean diameter shown on Figure 3 and Figure 4. The mean diameter in the lowest part of the riser is very high due to the presence of the larger particles. This accumulation of large particles gives the sharper gradient in the pressure drop seen on Figure 2.

Since the large particles do not circulate there will be a relatively larger amount of these particles in the riser compared to the particle size distribution of the whole system. If there are more large particles present in the riser there will be a much sharper gradient in the bottom of the riser than found in Case 1. To investigate how an altered distribution will influence the pressure profile the two cases with altered size distribution are run (Case 3 and 4). The resulting pressure profile can be seen on Figure 5.

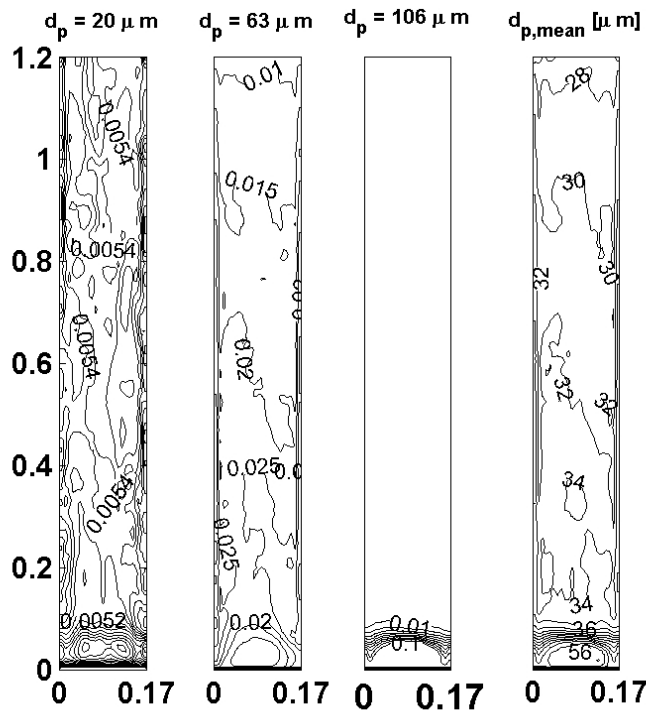


Figure 3. Plots of averaged solids volume fraction for the three particle sizes and average particle size.

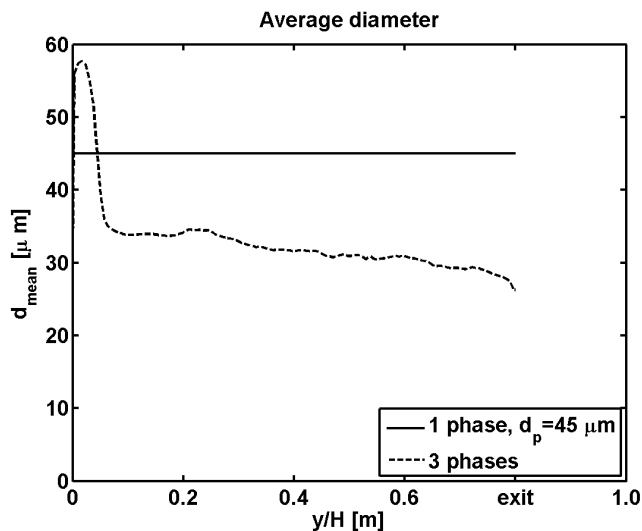


Figure 4. Average particle diameter as a function of height along the centerline of the riser.

The introduction of more of the large particles gives the expected effect. Since the larger particles are only present in the bottom part of the riser, the pressure gradient in this part of the riser is much sharper due to the increase in particles found here. The agreement with the experimental data reported by Ibsen *et al.* (144) is much better than in the case of having only one representative diameter.

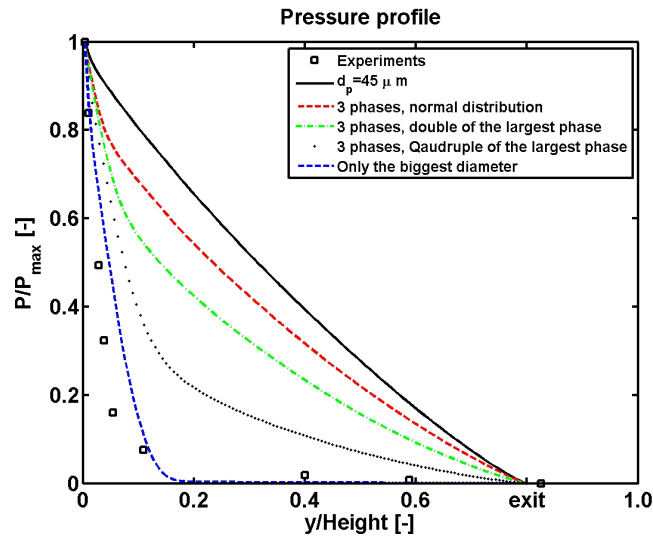


Figure 5. Comparison of simulated and measured pressure profile.

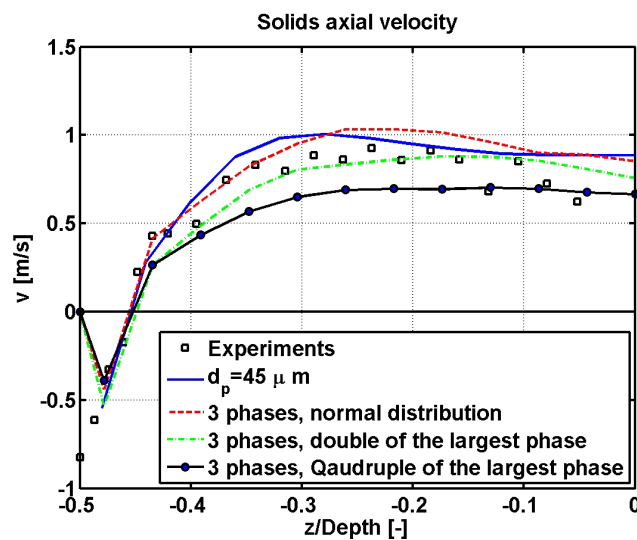


Figure 6. Comparison between measured a simulated solids axial velocity at  $y/H=0.5$ .

The best agreement is seen when using four times as many of the larger particles (Case 4). There is however still room for improvement which most likely to be found in the modeling of the inter-phase drag coefficient. There is work going on to improve this modeling such as Yang et al. (13) and van der Hoef et al (12), but this issue remains unsolved. From the two-fluid simulation using only large particles it is seen that the large particles indeed are only present in the bottom of the riser.

In the velocity profile seen on Figure 6 there is general agreement between the simulations and the experiments. There is a large gradient near the wall, and the profile is more flat in towards the center of the riser. As more large particles are introduced the average particle velocity is slightly reduced since the larger particles have a higher slip velocity.

## CONCLUSIONS

Multi-fluid CFD simulations of a CFB riser show that a mal-distribution of particles occurs in the CFB; the larger particles are retained in the riser, whereas the intermediate and small particles are distributed both in the return leg and the riser. Simulations using an altered particle size distribution show significant improvements in the pressure profile in the bottom part of the riser.

## REFERENCES

1. Agrawal, K., Loezos, P.N., Syamlal, M. and Sundaresan, S., "The role of meso-scale structures in rapid gas-solid flows", *J. Fluid Mech.*, 445, 151-185 (2001).
2. Deardorff, J.W., "On the magnitude of the subgrid scale eddy coefficient", *J. Comp. Phys.*, Vol. 7, 120-133 (1971).
3. Ding, J. and Gidaspow, D., "A Bubbling Fluidization Model using Kinetic Theory of Granular Flow. *AIChE Journal*", 36, 523-538 (1990).
4. Gibilaro, L.G., Di Felice, R. and Waldrum, S.P., "Generalized Friction Factor Coefficient Correlations for Fluid-Particle Interactions", *Chem. Eng. Sci.*, Vol. 40, No. 10, 1817-1823 (1985).
5. Glicksman, L.R., Hyre, M. and Woloshun, K., "Simplified scaling relationships for fluidized beds", *Powder Technology*, Vol. 77, 177-199 (1993).
6. Hansen, K.G., Ibsen C.H., Solberg T. and Hjertager B.H., "Eulerian/Eulerian CFD Simulation of a Cold Flowing FCC Riser", *International Journal of Chemical Reactor Engineering*, Vol. 1: A31 (2003).
7. Hansen, K.G., "A Three-Dimensional Numerical Study of Gas-Particle Flow and Chemical Reactions in Circulating Fluidized Bed Reactors". Ph.D. Thesis, Aalborg University Esbjerg, Denmark.
8. Ibsen, C.H., Solberg, T. and Hjertager, B.H., "Evaluation of a Three-dimensional Model of a Scaled Circulating Fluidized Bed", *Ind. Eng. Chem. Res.*, Vol. 40, 5081-5086 (2001).
9. Johnsson, F., Vrajer, A. and Leckner, B., "Solids flow pattern in the exit region of a CFB-furnace – Influence of exit geometry", in: R. Reuther (Ed.), *Fifteenth International Conference on Fluidised Bed Combustion*, ASME, New York, (1999).
10. Mathiesen, V.; Solberg, T. and Hjertager, B.H., "Prediction of Gas/Particle Flow With an Eulerian Model Including a Realistic Particle Size Distribution", *Powder Technology*, Vol. 112, 34-45 (2000).
11. Samuelsen, A.E. and Hjertager, B.H., "An Experimental and Numerical Study of Flow Patterns in a Circulating Fluidized Bed Reactor", *Int. J. Multiphase Flow*, Vol. 22, No. 3, 575-591 (1996).
12. van der Hoef, M.A., Beetstra R. and Kuipers, J.A.M., "Lattice Boltzmann simulations of low-Reynolds-number flow past mono- and bidisperse arrays of spheres: results for the permeability and drag force", *J. Fluid Mech.*, 528, 233–254 (2005).
13. Yang, N., Wang, W., Ge, W., Li, J., "CFD simulation of concurrent-up gas-solid flow in circulating fluidized beds with structure dependent drag coefficient", *Chem. Eng. Sci.*, 96, 71-80 (2003).
14. Zhang, D.Z. and VanderHeyden, W.B., "High-Resolution Three-Dimensional Numerical Simulation of Circulating Fluidized Beds", *Powder Tech.*, 116, 133-141 (2001).