INVESTIGATION OF THE DENSE FLOW REGIMES IN A GAS-SOLIDS FLUIDIZED BED WITH COMPLEX RISER SHAPE BY CFD SIMUALTION AND EXPERIMENT MEASURMENT

Haigang Wang[1,2*], Jiaimin Ye[1], Qiuya Tu[1], Xiaofang Wang[1]

[1]*Institute of Engineering Thermophysics, Chinese Academy of Sciences, Beijing, 100190, P.R. China*

[2] *University of Chinese Academy of Sciences, Beijing, 100190,P.R. China*

*Email: *wanghaigang@iet.cn*

Abstract – In this research, CFD simulations based on computational particle fluid dynamic (CPFD) and two-phase fluid model (TFM) are used to investigate the gas-solids flows in a circulating fluidized bed (CFB) system with M-shape riser and four cyclones to investigate the shape effect on the gas-solid flow hydrodynamics characteristics in the whole CFB system. Both TFM and CPFD simulation results in the dense flow regimes as well as in the looping system are given and compared with each other. Parts of the CFD simulation are compared and validated by pressure measurement. The flow hydrodynamics behavior with different models are given and analyzed based on the CFD simulation results. Key parameters including pressure and solids concentration, means particle concentration are given and compared with available experimental data. The research gives valuable information for the design of CFB system with multi-cyclone separators.

INTRODUCTION

Circulating fluidized bed (CFB) as one kind of clean coal combustion technology has been used in the power industry for many years due to its high heat and mass transfer efficiency. The current trend for the application of CFBs is scaling up from small scale to large scale with high thermal capacity. The gas-solid flows characteristics of is one of the key issues for a supercritical pressure CFB design and scale-up the CFB from small scale to large scale. However, due to the large capacity of combustion and multiple cyclones of the supercritical CFB boiler, the gas-solid flow is different from those of small capacity CFB boiler. In addition, fluidizing characteristics with large cross-sectional chamber and non-uniform distribution of gas-solid flow through multiple cyclones require further research (Jiang et al. 2014). To investigate the hydrodynamic behavior of circulating fluidized bed with complex combustion chamber, a cold CFB test rig with different number of cyclone separators on the top of the boiler and different riser shape were built in the Institute of Engineering Thermophysics, Chinese Academy of Sciences. The objective of this research is to investigate the gas-solids two phase flows characteristics in the CFB systems by CFD simulation and experimental measurement.

In the CFD simulation, the gas-solid flows in a scaling-down CFB with multi-cyclone separators was investigated using FLUENT/Barracuda platform and pressure drop measurement is used to validate the simulation results. To reduce the simulation time, a simplified CFB model with plenum chamber, air distributor, CFB combustion chamber and cyclone inlet duck, are used for the simulation. A simplified porous media model was used to simulate the pressure drop across the air distributor plate based the user-defined function (UDF) in FLUENT. Simulation results for the air and particles flows are given and compared. Key parameters including pressure-drop, velocity, solids phase concentration distribution in the risers are given and compared.

CFD MODEL

In this research, two approaches, namely Eulerian-Eulerian two-phase flows model (TFM) and Eulerian-Lagrangian based model computational particle fluid dynamic (CPFD), are used to simulate the gas-solids flow in the fluidized bed system as shown in Fig. 1. The details of two CFD approaches are given as follows.

TWO-FLUID MODEL

In this research, two CFD approaches are used to simulation the gas-solids two-phase flow in the CFB riser. The first approach is the Eulerian-Eulerian based two-fluid model (TFM) which is based on the conservation equations for mass and momentum for both the gas and solid phases. In the TFM treatment, both phases are considered to be continuous and fully interpenetrating, in contrast to the Eulerian-Lagrangian treatment which just used for the discrete phase model. Each phase has its own governing equations. All the relevant equations are given in references (Wang et al. 2006).

In this simulation, the gas is regarded as incompressible and the energy equation is neglected in the Navier-Stokes equations. There are no chemical reactions and mass transfer in the process. In the momentum equations, external body, lift, virtual mass forces are ignored, and the drag models used in this simulation is based on the Wen-Yu drag model (Wen and Yu, 1966). The solid phase pressure and stress is modeled based on the kinetic granular theory (Gidaspow et al., 1992). The solid pressure is composed of a kinetic term and a second term due to granular collision (Lun et al., 1984). The solids stress tensor contains shear and bulk viscosities arising from particle momentum exchange due to translation and collision. The gas-solid flow progress in the fluidized bed was assumed as turbulence-free flow, ignoring the turbulence of the phases.

CPFD MODEL

Computational particle fluid dynamic (CPFD) is an Eulerian-Lagrangian approach which solves the gas solid flow in three dimensions. The continuous fluid phase is calculated based on the Navier-Stokes equations, discrete solids phase is based on Multiphase Particle-in-cell (MP-PIC) numerical method and these two are coupled by interphase drag force (Andrews 1996, Snider 2001). CPFD can be used for industry scale models with large number of particles due to its numerical parcels concept, which is a numerical approximation, contains quite a number of physical particles with same properties in the same computational cell. The details for the CPFD approach and conservation equations for the gas phase and parcel treatment are given in reference in details (Jiang et al. 2014 and Qiu et al. 2015).

In the CPFD simulation, the Wen-Yu/Ergun drag model is used to simulate the gas-solids interface. The model is given as

$$D_p = C_d \frac{3}{8} \frac{\rho_g}{\rho_p} \frac{\left| v_g - v_p \right|}{\left(3V_p / 4\pi \right)^{1/3}} \tag{1}$$

where C_d is the drag coefficient. It depends on the Reynolds number, i.e.

$$C_d = \frac{24}{\mathrm{Re}} \left(1 + 0.15\,\mathrm{Re}^{0.687} \right) \theta_g^{-2.65} \quad \text{for} \quad \mathrm{Re} < 1000 \tag{2}$$

$$C_d = 0.44\,\theta_g^{-2.65} \qquad \text{for} \ \mathrm{Re} \geq 1000 \tag{3}$$

The Reynolds number is given by

$$\mathrm{Re} = \frac{2\rho_g \left| v_g - v_p \right|}{\mu_g} \left(\frac{3V_p}{4\pi} \right)^{1/3} \tag{4}$$

where ρ_g is the gas density, μ_g is the gas viscosity, v_g and v_p represent the gas and particle velocity respectively, and V_p is the volume of a particle.

BOUNDARY CONDITIONS

To carry out the simulation, the boundary conditions of gas phase and particle phase for the TFM simulation at the inlet, wall and outlet of the CFB risers are specified and listed in Table 1. The boundary conditions for the CPFD simulation are given in Table 2. The wall of the fluidized bed is treated as non-slip boundary for the air phase, i.e. the velocity of the air phase is set to zero at the wall. For the solids phase the free slip condition is assumed, i.e. there is no hindrance in the downward or upward velocity of the particles when they are near the wall. At the outlet, the pressure of ambient atmosphere and continuous mass fluxes of both air and solids phase are assumed. For the particle simualtion in the CPFD approach, the particle diameter distributor model of Rosin-Rammler is used with the median diameter of 400 μm and dispersion index of 4.47. In the TFM approach, the mean diameter of 400 μm are used for the CFD simulaiton.

Table 1. Doundary conditions for the TFM simulation

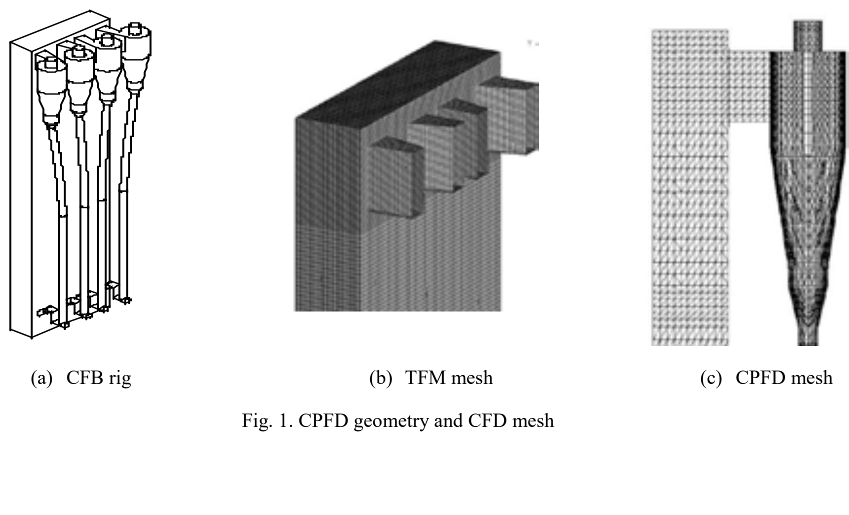
Boundary	Gas phase	Particle
Inlet	Velocity: 5 m/s	Velocity: 5 m/s
Wall	No-slip boundary	Restitution coefficient for tangental: e=0.89 Restitution coefficient for normal: e=0.63
Outlet	Fully developed	Fully developed

Table 2 Input parameters in the CPFD simulation.

Particle-to-wall interaction	Normal retention coefficient, e_n	0.89
	Tangential retention coefficient, e_t	0.68
	Diffuse bounce, D_f	0
Solver settings	Time step, Δt	4E-4 s
	Total time	32 s
	gravitational acceleration, g	-9.81
	Maximum volume iteration	1
	Volume residual	1.00E-05
	Maximum pressure iterations	2000
	Pressure residual	1.00E-08
	Maximum velocity iterations	50
	Velocity residual	1.00E-07
	Maximum momentum redirection from collision	40%
	close pack fraction limit	0.65

CFD-CPFD MESH

Fig. 1 shows the scaled down CFB models from a 300-MWe supercritical CFB boiler under the scale of 1:20. To achieve high solid separation efficiency, four cyclone separators and U-shape loop seal are used in the system. Fig. 1 (b) gives mesh for TFM simulation and (c) for the CPFD simulation. The computational domain is divided into several blocks to increase calculation accuracy with partial improvement of grid density in the cyclone and air distributor plate in the TFM simulation. The geometric model is meshed by Cartesian grid and the total number of grid cell is around 200,000~300,000 for the TFM simulation and 100,000 cells for CPFD simulation. The minimum volume of grid size is 1.09×10^{-9} cm^3 which is sufficient for the simulation convergence (Solnordal et al. 2015). The simulation process with all described assumptions and boundary conditions were simulated with the BARRACUDA-CPFD® and FLUENT packages respectively.



(a) CFB rig (b) TFM mesh (c) CPFD mesh

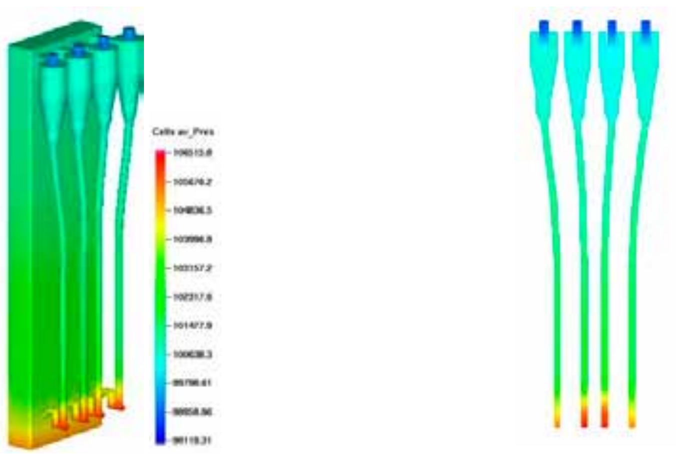
Fig. 1. CPFD geometry and CFD mesh

RESULTS AND ANALYSIS

Both in the TFM and CPFD simulations, the first 10 seconds were neglected to disregard the effect of the impulsive initialization effect. The real simulation times for both cases are longer than 20 seconds to ensure a complete circulation of particles. The following parts will give the TFM-CPFD simulation results in terms of pressure, solid volume fraction, velocity and solid recirculation flow rate through the four cyclones.

DISTRIBUTION OF PRESSURE

Fig. 2 gives the pressure distribution in the whole CFB system and four cyclones respectively. From Fig. 1 (a), it can be seen that the pressure drops in the bottom region and in the cyclone are much higher than that in other flow region. Fig. 3 compares the CPFD simulation results for the pressure drop along the riser with measurement. In the experiment, the measured points for pressure are located at five layers in the vertical direction in the CFB riser. The area-weighted pressures taken from different heights along the CFB riser obtained by simulation are compared with measurements. From the comparison result as shown in Fig. 3, it can be found out that CPFD has a higher pressure drop prediction both for the dilute and dense regions in the riser, which will result in the overestimation of solids volume fraction in these regions.



(a) 3D distibution (b) In cyclones

Fig. 2. Pressure distribution from CPFD simulation

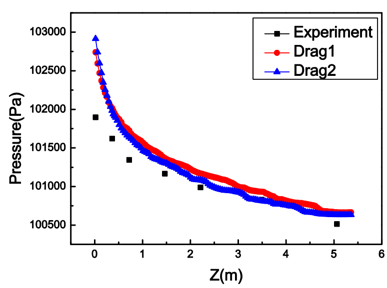
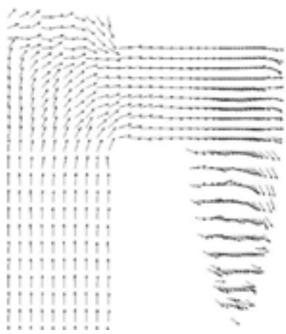


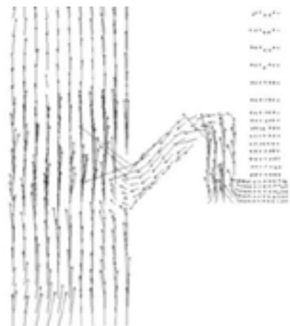
Fig. 3. Comparison between CPFD simulation and measurements

DISTRIBUTION OF VELOCITY

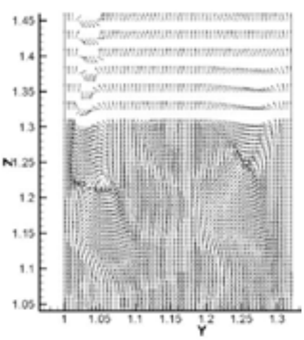
Fig. 4 shows the velocity field of gas phase at the top and the bottom of the CFB riser. From Fig. 4 (a), it can be found out that the gas flow changes its direction from vertical to horizontal as it approaches the inlet of cyclones. The gas flow has a strong rotation in the center of cyclone, which can also be observed in Fig. 4 (d). The gas flow tends to flow downward along the wall of cyclone and then turns around joining the central upward flow right below the outlet of cyclone until it flows through the outlet. In the bottom of the CFB riser, the gas flow has non-uniform distribution with both upward and downward flows as shown in Fig. 4 (c). From Fig. 4 (b), it can also be observed that the air flow used to fluidize the particles in the U-loop seal flows back into the CFB riser and particles in the standpipe are dominated by the gravitational force.



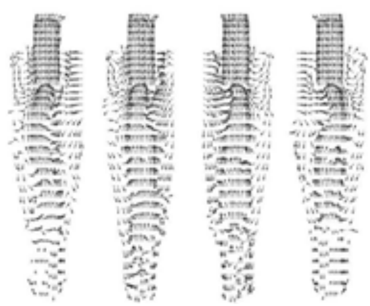
(a) In the top of the riser



(b) In the bottom of the riser



(c) In the bottom of the riser



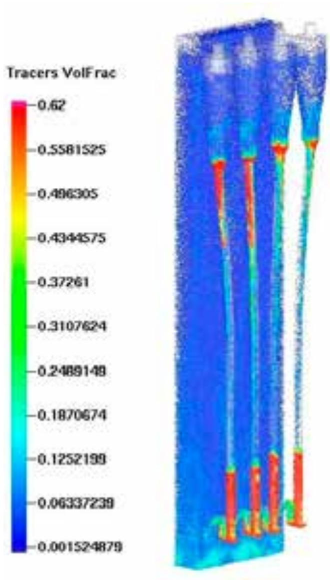
(d) In cyclones

Fig. 4 Velocity vector field in different regions

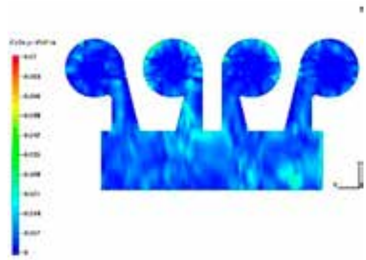
DISTRIBUTION OF SOLID CONCENTRATION

In order to avoid the impulsive initialization effect and ensure the circulation reaching stability, in the total 30s simulation, only the last 10 s data was used for the analysis. Fig. 5 shows several instantaneous particle volume fractions in the riser and cross-section of the cyclones. In the bottom region below 1 m, the solids volume fraction is relatively large, which shows that this is the dense region containing particles with larger diameters. Small particles circulate in the riser or entering into the recirculation system. In the top of the CFB riser, it can be observed that particles move from the central area to the outside wall which means internal circulation exists in the cross section of the combustion chamber. From the concentration as shown in Fig. 5 (b), it can be observed that the particle concentration near the inner ring wall is lower than that of the outside wall in the cyclone inlet duck. Fig. 6 compares the particle volume fraction in the bottom of the

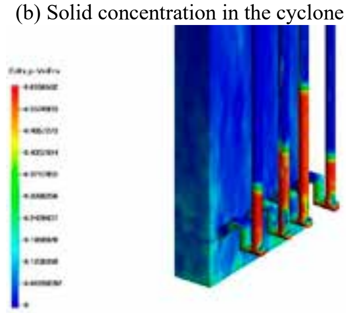
CFB riser predicted by TFM and CPFD simulations. Both of the simulations can give the high concentration in the bottom region and non-uniform distribution for the solids phase in the CFB riser.



(a) 3D solid concentration

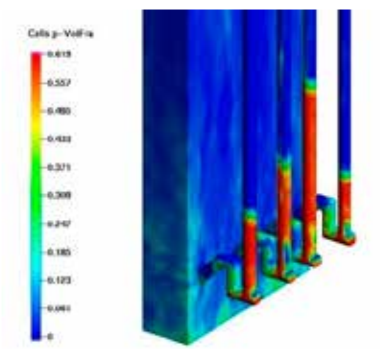


(b) Solid concentration in the cyclone

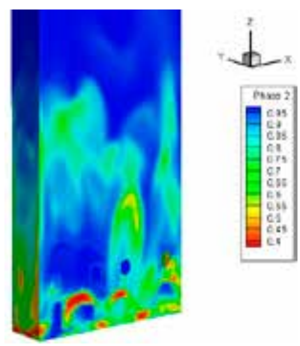


(e) Solid concentration in the bottom and U-loop

Fig. 5 Solid concentration by CPFD simulation



(a) CPFD simulation



(b) TFM simulation

Fig. 6 Solid concentration from CPFD and TFM simulation

Fig. 7 gives the time-averaged solids concentration from TFM-CPFD simulations and pressure drop predictions in the CFB riser. As can be seen from Fig. 7, both TFM and CPFD predict a bottom-dense and upper-dilute structure for the solids concentration. In both cases, the simulation results are in well agreement with experimental data in the region of z>1 m. However, in the dense region, the CPFD predictions for solids concentration are smaller than the experiment data in the region below z=1 m, which means more particles are brought upwards and thus the predictions for upper region are denser than the experiment. Alternatively, the TFM predictions for the solid concentration are higher than the experiment result in this region. This is mainly due to the inaccurate estimation of drag force in the dense region for the CFD simulations.

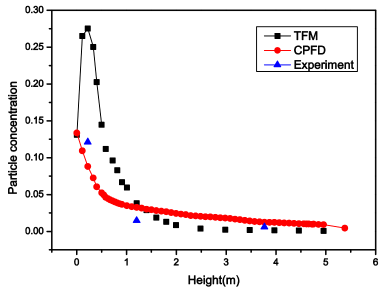
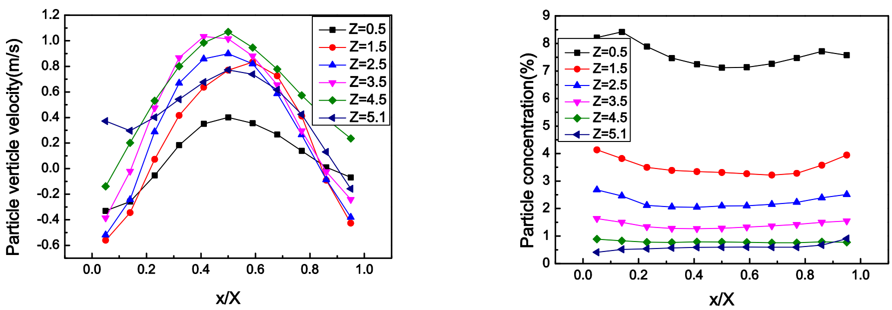


Fig. 7 Solid concentration profiles

Fig. 8 shows the horizontal distribution of particle vertical velocity and concentration at different heights along the riser from the CPFD simulation. The simulation result shows that the particle velocity is higher in center. In contrast to the core upward flow, the particles form a downward flow near the wall. This is the core-annulus structure or "crown-like" shape concentration which is congruent with the results reported from the TEM simulations (Malcus *et al*. 2002, Wang *et al*. 2006). In the upper region of the riser, two peaks instead of one are observed in the distribution curve at z=5.5 m, which may be caused by the existence of horizontal flows through cyclone inlets.



(a) Velocity (b) Concentration

Fig. 8 particle velocity and concentration along the height of the riser

PARTICLE RE-CIRCULATION FLUX

Particle re-circulation flux into the riser from the U-loop is an important parameter to indicate the performance of a CFB boiler. As the particle mass flow rate at the inlet of the cyclones fluctuates rather intensely over time, a time-averaged estimation of the flux is taken after the gas-solid flow become steady. Fig. 9 gives the particle flux distribution among the four cyclones from the TFM-CPFD simulation and experimental measurement.

From the results it can be seen that the cyclone in the middle has a high particle mass flow rate than the rest. It reveals that more particles go through cyclones located at the center. The total mass flow rates in both sides are almost the same, which means that the distribution of particles in the two sides of the riser is quite uniform. However, the CPFD result is higher than the experimental result and the TFM simulation gives a lower value compared with the measurements. As discussed in the previous part, the main reason is that the CPFD overestimates the drag force and TFM model underestimate the drag force which result in the prediction errors for the particles phase flows. Further research is necessary to address the drag model effect on the prediction accuracy both in TFM and CPFD model.

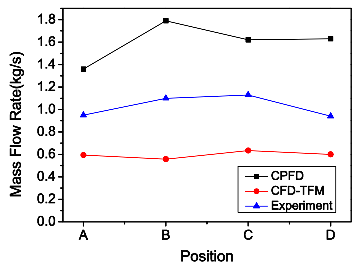


Fig. 9 Solid mass flow rate through the four cyclones

CONCLUSIONS

The gas-solids flow characteristics in a CFB system with four cyclone separators is investigated based on TFM-CPFD simulation and pressure measurement. The gas flow is relatively uniformly in the CFB riser. The pressure drops of cyclones and dense region account for a large proportion of the whole pressure drop. The particle phase is non-uniform among the four parallel cyclones. The particle concentrations of four cyclones in the center are higher than that of the others. Future work should be addressed on drag model effect on the gas-solid inter-force predictions.

ACKNOWLEDGEMENTS

The authors are grateful to the support from the National Natural Science Foundation of China (No. 61320106004) and the Strategic Priority Research Program of the Chinese Academy of Sciences (No. XDA07030100). The authors would also like to thank CAS Interdisciplinary Innovation Team for supporting this research.

REFERENCES

Andrews, M.J., O′ Rourke, P.J. 1996. The multiphase particle-in-cell (MP-PIC) method for dense particulate flows. International Journal of Multiphase Flow 22 379-402.

Jiang, Y., Qiu, G.Z., Wang, H.G. 2014. Modelling and experimental investigation of the full-loop gas-solid flow in a circulating fluidized bed with six cyclone separators. Chemical Engineering Science 109 85-97.

Gidaspow, D., Bezburuah, R., Ding, J. 1992. Hydrodynamics of Circulating fluidized bed, Kinetic theory approach. In Fluidization VII. *Proc. of 7th Eng. Foundation Conference on Fluidization*, Brisbane, Australia, May 3-8, 1992;75-82.

Lun. C.K.K., Savage. S.B., Jeffrey. D.J., Chepurniy, N. 1984. Kinetic theories for granular flow: inelastic particles in coquette flow and slightly inelastic particles in a general flow field. Journal of Fluid Mechanical 140 223-256.

Malcus, S., Cruz, E., Rowe, C., Pugsley, T.S. 2002. Radial solid mass flux profiles in a high-suspension density circulating fluidized bed. Powder Technology 125 5-9.

Qiu, G.Z., Ye, J.M., Wang, H.G. 2015. Investigation of gas-solids flow characteristics in a circulating fluidized bed with annular combustion chamber by pressure measurements and CPFD simulation. Chemical Engineering Sciences 134 433-447.

Snider, D.M. 2001. An incompressible three-dimensional multiphase particle-in-cell method for dense particle flows. Journal of Computational Physics 170 523-549.

Solnordal, C.B., Kenche, V., Hadley, T.D., Feng, Y.Q., Witt, P.J., Lim, K.S. 2015. Simulation of an internally circulating fluidized bed using a multiphase particle-in-cell method. Powder Technology 274 123-134.

Wang, H.G., Yang, W.Q., Dyakowski, T., Liu S. 2006, Study of bubbling and slugging fluidized beds by simulation and ECT, AIChE Journal 52 3078-3087.

Wen, C.Y., Yu, Y.H. 1966. Mechanics of fluidization, Chemical Engineering Progress Symopsium Series. 62 100-111.