Ninth International Conference on CFD in the Minerals and Process Industries
CSIRO, Melbourne, Australia
10-12 December 2012

A NOVEL CIRCULATING FLUIDIZED BED TO IMPROVE FLUID-SOLIDS CONTACTING

K.W. CHU* **and A.B. YU**

Lab for Simulation and Modelling of Particulate Systems, School of Materials Science and Engineering,
The University of New South Wales, Sydney, NSW 2052, Australia
*Corresponding author, E-mail address: kaiwei.chu@unsw.edu.au

ABSTRACT

It has been widely reported that traditional circulating fluidized beds (CFBs) suffer from core-annulus flow structure which is characterized by severe solids back-mixing and non-uniform distribution of solids in both axial and radial directions. The unfavorable core-annulus flow structure in the riser reactors has led to considerations of new schemes of contacting for a refinery of the 21st Centry (Gidaspow et al., 2004). In this work, an S-shaped design of risers is proposed and its effectiveness is examined first by numerical simulations then by physical experiments. Both simulations and experiments show that the flow pattern in the S-shaped risers is significantly different from that in traditional straight risers. Generally speaking, the uniformity of the distribution of solid concentration and velocity is improved in the "S-shaped" risers, with the core-annulus flow structure of a traditional design almost gone. The reason that S-shaped designs differs is that the high fluid velocity regions move back and forth in the system which keeps the boundary layer of solids from being established. This S-shaped design may be also applicable to other vertical solids-fluid flow systems.

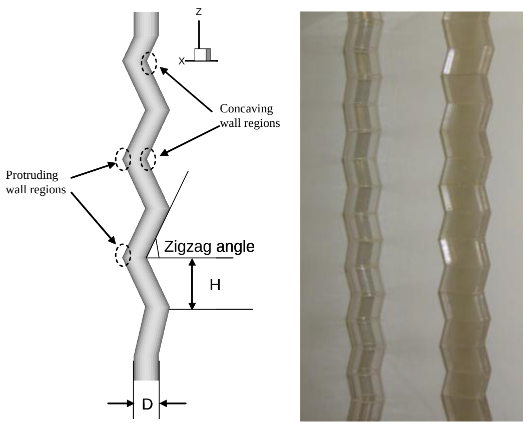
Keywords: solids-fluid flow, circulating fluidized bed, conveying, computational fluid dynamics, discrete element method.

INTRODUCTION

Circulating fluidized beds (CFBs) are applied widely in chemical process and energy conversion industries. They are used as thermal reactors such as combustors, boilers and gasifiers, in coal, power and hazardous waste elimination industries. It is a key component (where most of the cracking happens) in a fluid catalytic cracking (FCC) process in petroleum refineries to convert heavy oil to light products. Some of the new applications include synthesis of olefins from methanol, CO2 capture by solid sorbents, and coal and biomass gasification.

It is found that CFB process suffers many problems while it is ideal for solids-fluid contacting while compared with bubbling fluidized bed. Core-annulus flow structure is the most well-known one and has been extensively reported in the literature. It is characterized by the fact 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 (Subbarao and Basu, 1986 ; Grace, 1990 ; Rhodes et al., 1998). Core-annulus flow structure leads to insufficient gas-solid contact and back-mixing of solids (van Deemter, 1961) and thus non-uniform distribution of temperature and/or high temperature spot in boilers. It is even a more critical issue in the riser of a CFB unit in a FCC process in petroleum industry. The oil industry discovered this phenomenon only after publication of a paper by Sinclair and Jackson (1989) and measurements using gamma ray densitometers (Sun and Koves, 1998) by which it was learned that their large diameter risers operate in the core-annular regime: the core is very dilute, which means there may be no reactions in the core. Core-annulus flow structure causes insufficient gas-solid contacts and back-mixing of solids. Such issues have led to the consideration of new schemes for improving the gas-solid contacts in the refineries of the 21st Century (Gidaspow et al., 2004).

Previously, the core-annulus flow problem has been addressed by some researchers. The most popular solution is the development of the innovative downer reactor where solids and fluid are fed from the top of the reactor and the gas and solids flow downward (i.e., in the direction of gravity) (Zhu et al., 1995 ; Deng et al., 2002 ; Zhang et al., 2008). Another solution is the use of wall baffles inside the riser (Chen, 2006).



(a) (b)

Figure 1. Schematic representation (D=0.05 m) of a typical "S-shaped" fluid-particle flow system: (a), schematic drawing; and (b), experimental photo.

In this work, an innovative CFB system, which is named "S-shaped" CFB and shown in Figure 1, is developed to overcome the core-annulus problem in the riser reactors in petroleum industry. The performance of the S-shaped riser was investigated both by simulations and experiments.

MODEL DESCRIPTION

The numerical model used is a combined computational fluid dynamics and discrete element method (CFD-DEM) (Tsuji et al., 1993 ; Xu and Yu, 1997 ; Yu and Xu, 2003). In the CFD-DEM model, the motion of particles is modeled as a discrete phase, described by the Newton's laws of motion on an individual particle scale, while the flow of fluid (gas or liquid) is treated as a continuum phase, described by the local averaged Navier-Stokes equations on a computational cell scale. The method has been recognized as an effective method to study the fundamentals of particle-fluid flows under different conditions, as briefly reviewed by various investigators (Yu and Xu, 2003 ; Feng and Yu, 2004 ; Deen et al., 2007 ; Zhu et al., 2007,2008). CFD-DEM, as a method for modeling particle-fluid flows, has been well documented in the literature (e.g., (Cundall and Strack, 1979 ; Xu and Yu, 1997 ; Yu and Xu, 2003 ; Feng and Yu, 2004 ; Deen et al., 2007 ; Zhou et al., 2010)). For the completeness, here we just gave a very brief description of the method.

In current CFD-DEM model, the solid phase is treated as a discrete phase and described by DEM (1979). According to the model, the translational and rotational motions of a particle in a system at any time, t, can be described by Newton's second law of motion:

$$m_i \frac{d\mathbf{v}_i}{dt} = \mathbf{f}_{pf,i} + \sum_{j=1}^{k_i} \left(\mathbf{f}_{c,ij} + \mathbf{f}_{d,ij} \right) + m_i \mathbf{g} \tag{1}$$

and

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

The governing equations of gas phase are the same as those used in the well established two-fluid model (TFM) (Anderson and Jackson, 1967 ; Gidaspow, 1994 ; Enwald et al., 1996). They are thus the conservations of mass and momentum in terms of the local mean variables over a computational cell, given by

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

and

$$\frac{\partial (\rho_f \varepsilon \mathbf{u})}{\partial t} + \nabla \cdot (\rho_f \varepsilon \mathbf{u}\mathbf{u}) = -\nabla \mathbf{p} - \mathbf{F}_{pf} + \nabla \cdot (\varepsilon \boldsymbol{\tau}) + \rho_f \varepsilon \mathbf{g} \tag{4}$$

where $\mathbf{F}_{pf}$ is volumetric particle-fluid interaction force ($\mathbf{F}_{fp} = \sum_{i=1}^{k_c} \mathbf{f}_{pf,i}$, where k_c is the number of particles in a CFD cell and $\mathbf{f}_{pf,i}$ is the sum of particle-fluid interaction forces acting on particle i). In this model, the particle-fluid interaction force includes viscous drag force ($\mathbf{f}_{d,i}$) and pressure gradient force (PGF, $\mathbf{f}_{pg,i}$). The meaning of the other symbols and the equations used to calculate the forces in Eqs. (1-4) can be found elsewhere (Xu and Yu, 1997 ; Zhou et al., 2004 ; Chu et al., 2011).

SIMULATION AND EXPERIMENT CONDITIONS

Both numerical and physical experiments were conducted to investigate the performance of the S-shaped fluid-particle flow system. Numerical simulations were firstly carried out for different S-shaped designs to find out the optimum design. Then the optimum design was used in the physical experiments.

Generally speaking, as shown in Figure 1 and Figure 2, there are three parameters to be used to design an S-shaped fluid-particle flow system: the first one is the zigzag angle; another one is the ratio of H/D where D is the characteristic diameter of the cross section if the cross section is not a circle or square; and the final one is the shape of cross section which can be circle, square or other polygon. Three different designs were used in the simulation. Design I is with zigzag angle of 75 degree, H/D ratio of 2 and a circle cross section. Design II is with zigzag angle of 82 degree, H/D ratio of 2 and a circle cross section. Design III is with zigzag angle of 82 degree, H/D ratio of 1 and a square cross section and shown in Figure 2. Design III is though as the best of the three designs according to simulation results and thus used in the experiments. The value of D is 10 mm for all of the three designs in the simulation. In the experiments, one S-shaped design and one conventional straight design were used and their total height are both 1m. The S-shaped design in experiments is a five times scaled model of the S-shaped Design III (D = 50mm). The conventional straight design in experiments is in two-dimensional with width of 50 mm and thickness of 5 mm.

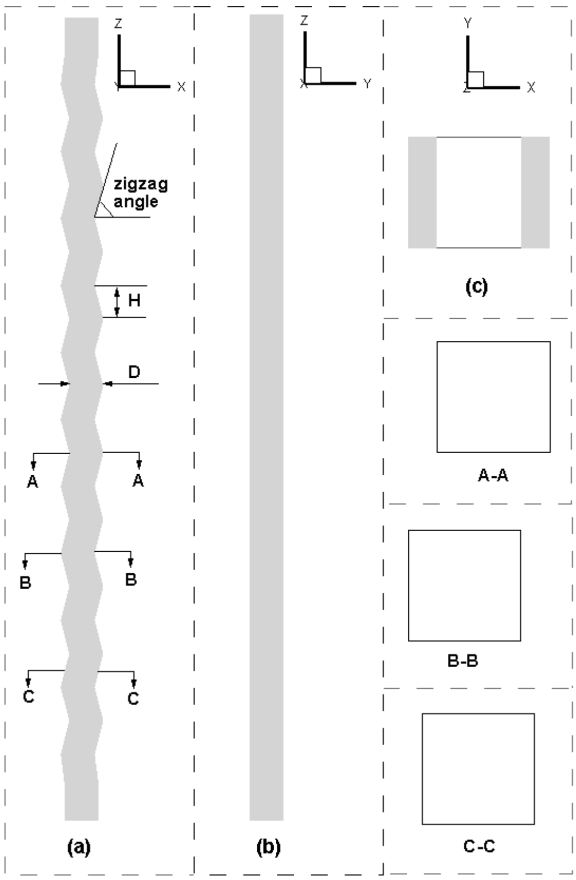


Figure 2. An optimized design of the S-shaped CFB in this work. (a), XZ view; (b), YZ view; (c), XY view.

Table 1. Parameters used in the simulations

Phases	Name	Symbol	Unit	Value
Particle phase	Density	ρ	Kg/m^3	1000
	Particle radius	R_i	mm	0.14; 0.20; 0.25
	Particle-particle and particle-wall rolling friction coefficient	μ_r	mm	0.005
	Particle-particle and particle-wall sliding friction coefficient	μ_s	dimensionless	0.3
	Poisson's ratio	ν	dimensionless	0.3
	Young's modulus	E	N/m^2	1×10^7
	Damping coefficient	c	dimensionless	0.3
	Time step	t	s	1×10^{-6}
Gas phase	Density	ρ	Kg/m^3	1.225
	Viscosity	μ	Kg/m/s	1.8×10^{-5}
	Velocity (m/s)	v	m/s	10.0
	Time step (s)	t	s	1×10^{-5}

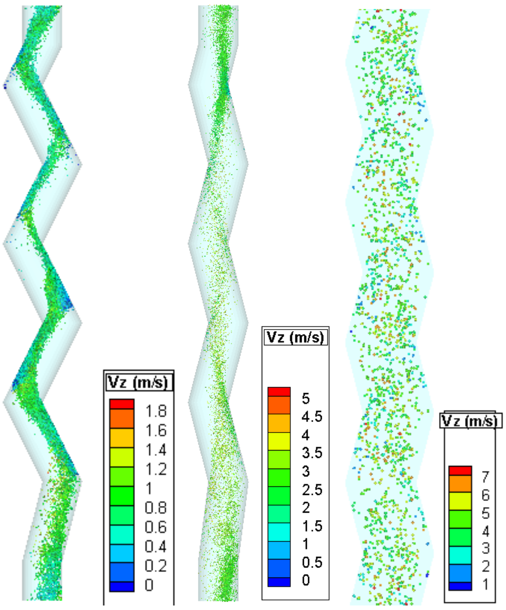
The operation conditions in the simulation and experiments are shown in Tables 1 and 2 respectively. In the simulation, in order to save computational cost, coarse particles were used. For the simulation of gas phase, non slip boundary condition is applied and k-ε turbulence model in Fluent is used for fluid phase. In the experiments, various gas velocities and particle sizes are used but here we only report the results of fine particles since the particle phase consists of fine particles in most of the industrial fluid-particle flow systems especially in riser reactors.

Table 2. Parameters used in the experiments.

Phases	Name	Symbol	Unit	Value
Particle phase	Type	--	--	Glass beads
	Density	ρ	Kg/m^3	2500
	Particle radius	R_i	mm	0.04-0.07
Gas phase	Type	--	--	Air
	Density	ρ	Kg/m^3	1.225
	Viscosity	μ	Kg/m/s	1.8×10^{-5}
	Velocity (m/s)	v	m/s	10.0

RESULTS

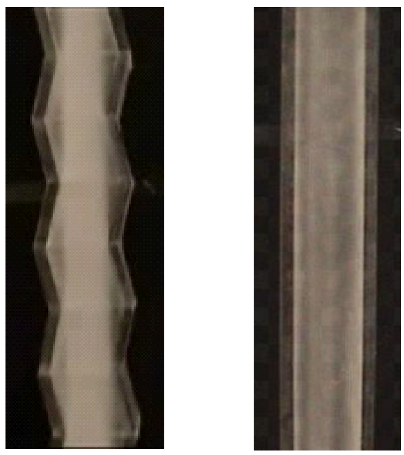
Figure 3 shows the simulated flow structures of three different designs of the S-shaped gas-particle flow system. In all three designs, the velocity of particles in the axial z-direction is larger or equal to zero, and the core-annulus flow structure is not observed here. From Figure 3(a) it can be clearly seen that the flow structure in an S-shaped design are quite different from that in a conventional straight design. Here, particles flow from one side of the system to the other periodically. One obvious shortcoming of the flow in Design I is that the particle concentration is not uniform in the system. In Design II, as a result of the increased zigzag angle, the particles mainly flow along the centre of the system, and the particle concentration is more uniform than in Design I. In design III, where H/D ratio is one, the flow is much more uniform than in Designs I and II. According to the simulation results, the design III of Figure 3 is considered as an optimized design. Thus, the Design III with its values of the zigzag angle and H/D ratio was chosen for the experimental study.



(a) (b) (c)

Figure 3. Numerical results showing macroscopic flow structures of three different designs of S-shaped gas-particle flow system: (a), S-shaped Design I; (b), S-shaped Design II; and (c), S-shaped Design III.

The difference of the flow structures between an S-shaped and a conventional straight design is shown in Figure 4. Glass beads with the sizes ranging from 0.04 to 0.07 mm were used in both cases of Figure 4. As shown in Figure 4(a), the flow can be considered as close to an ideal flow: the particles are mainly in the centre of the system, where the particle concentration is largely uniform. There are very few particles in the protruding regions of the system. It looks as if there is a vertical inner cylinder inside the system, formed by particles. Moreover, it is observed that all the particles flow upward in the S-shaped system, and therefore, there is no build-up of solids for back-mixing. However, in a conventional straight design, as shown in Figure 4(b), there is a core-annulus flow structure: particles are mainly close to the walls of the system, with less particles flowing in the middle. Although a quantitative comparison should be carried out to investigate the difference between an S-shaped design and a conventional straight design, the current experimental results clearly show that the S-shaped design is quite promising.



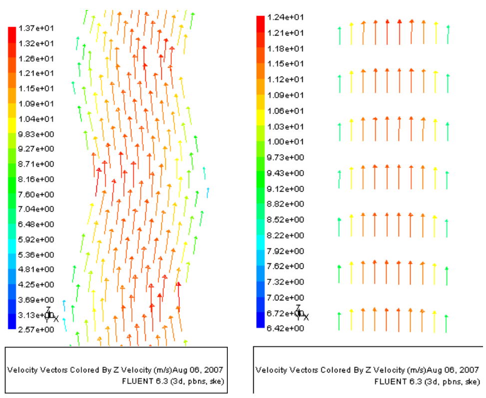
(a) (b)

Figure 4. Comparison of experimentally obtained flow structures: (a), S-shaped design; and (b), straight design.

The reason why an S-shaped design works better than a conventional straight design could be explained with reference to Figure 5. In an S-shaped design, the high fluid velocity region shifts from one wall side to the opposite periodically. This sideways motion of the high fluid velocity regions keeps the wall layers of particles from being established, and thus, it is difficult for solid back-mixing to occur along the wall in the S-shaped system.

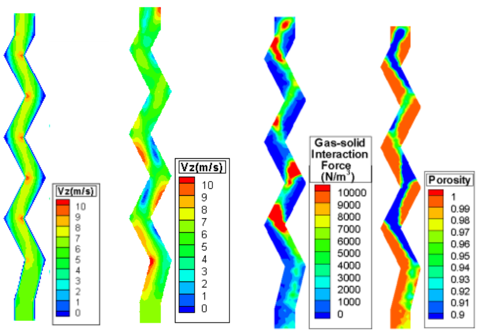
However, in a conventional straight system, as shown in Figure 5(b), there is no sideways motion of high fluid velocity regions, and the fluid velocity is always higher in the centre but lower in the regions close to the wall because of wall effect. Thus, in this case, once particles are introduced into the system, the particles near the wall will move upward slower than those near the centre because the fluid velocity near the walls is lower than that in the middle. Consequently, there will be more particles accumulating the near wall regions than in the central region. When more particles collect near the walls, fluid near the walls will face the increased volumetric fluid-particle interaction force, which will make more fluid to pass through the central region. The larger the extent of fluid flow through the central region, the larger the accumulation of particles near the wall. Finally, the core-annulus flow structure is established in the conventional straight fluid-particle flow system (Chu and Yu, 2008).

From the above analysis, it is clear that the core-annulus flow structure problem is initiated by the wall effect and intensified by non-uniform distribution of the gas-particle volumetric interaction force. Indeed, gas-particle interaction can be quite strong in a gas-particle flow system. Figure 6 shows a typical example of the effect of fluid-particle interaction. It can be seen from Figure 6 (a) and (b) that the gas flow greatly changes due to the loading of solids. Under the gas-only flow condition, the high gas velocity regions are near the concaving wall regions (see Fig. 1) of the S-shaped riser. However, under gas-solids flow condition, the high gas velocity regions are near the protruding wall regions. The change in the fluid velocity field is thought to be caused by the volumetric gas-particles interaction force, which is shown in Figure 6 (c). It shows that fluid tends to flow through where the resistance force is relatively smaller. Figure 6 (d) clearly shows that the high solids concentration regions shift from right-side wall to left-side wall periodically, which is quite different from the flow structure in a traditional straight bed.



(a) (b)

Figure 5. Comparison of fluid velocity distributions in the condition of pure gas flow: (a), S-shaped design III; and (b), conventional straight design.



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

Figure 6. The effect of solids flow on the gas field in the S-shaped Design I: (a), gas velocity of pure gas flow; (b), gas velocity of gas-solids flow; (c), time-averaged porosity over 1.3s; and (d), volumetric gas-solids interaction force at t=1.3s.

CONCLUSION

A novel fluid-particle flow system has been developed and investigated both numerically and experimentally. It is shown that the flow structure in the S-shaped designs is obvious different from that in conventional straight designs. In the optimized S-shaped design in this work, the particle concentration is higher in the centre of the system and lower near wall regions. The reason is that the high fluid velocity regions move back and forth in the system which keeps the boundary layer of solids from being established. The developed S-shaped fluid-particle flow system shows a high potential to solve the core-annulus flow structure problem suffered in conventional straight fluid-particle flow systems especially in riser reactors of petroleum industry. Quantitative and extensive work to compare the performance of S-shaped designs with conventional straight designs is necessary to further verify the usefulness of the S-shaped fluid-particle flow systems.

ACKNOWLEDGEMENT

The authors are grateful for the financial support from the Australian Research Council.

REFERENCES

ANDERSON T. B., JACKSON R., (1967). "A fluid mechanical description of fluidized beds". Industrial & Engineering Chemistry Fundamentals, 6, 527-39.

CHEN Y. M., (2006). "Recent advances in FCC technology". Powder Technology, 163, 2-8.

CHU K. W., WANG B., YU A. B., XU D. L., CHEN Y. X., (2011). "CFD-DEM simulation of the gas-solid flow in a cyclone separator". Chemical Engineering Science, 66, 834-847.

CHU K. W., YU A. B., (2008). "Numerical simulation of complex particle-fluid flows". Powder Technology, 179, 104-114.

CUNDALL P. A., STRACK O. D. L., (1979). "Discrete numerical-model for granular assemblies". Geotechnique, 29, 47-65.

DEEN N. G., ANNALAND M. V. S., VAN DER HOEF M. A., KUIPERS J. A. M., (2007). "Review of discrete particle modeling of fluidized beds". Chemical Engineering Science, 62, 28-44.

DENG R., WEI F., LIU T., JIN Y., (2002). "Radial behavior in riser and downer during the FCC process". Chemical Engineering and Processing, 41, 259-266.

ENWALD H., PEIRANO E., ALMSTEDT A. E., (1996). "Eulerian two-phase flow theory applied to fluidization". International Journal of Multiphase Flow, 22, 21-66.

FENG Y. Q., YU A. B., (2004). "Assessment of model formulations in the discrete particle simulation of gas-solid flow". Industrial & Engineering Chemistry Research, 43, 8378-8390.

GIDASPOW D. (1994). Multiphase Flow and Fluidization: Academic Press.

GIDASPOW D., JUNG J., SINGH R. K., (2004). "Hydrodynamics of fluidization using kinetic theory: an emerging paradigm: 2002 Flour-Daniel lecture". Powder Technology, 148, 123-141.

GRACE J. R., (1990). "High-velocity fluidized-bed reactors". Chemical Engineering Science, 45, 1953-1966.

RHODES M. J., SOLLAART M., WANG X. S., (1998). "Flow structure in a fast fluid bed". Powder Technology, 99, 194-200.

SINCLAIR J. L., JACKSON R., (1989). "Gas-particle flow in a vertical pipe with particle-particle interactions". AIChE Journal, 35, 1473-1486.

SUBBARAO D., BASU P., (1986). "A model for heat-transfer in circulationg fluidized beds". International Journal of Heat and Mass Transfer, 29, 487-489.

SUN B., KOVES W. J., (1998). "Application of a numerical hydrodynamic model in FCC design". AIChE Symposium Series, vol.II, 469.

TSUJI Y., KAWAGUCHI T., TANAKA T., (1993). "Discrete Particle Simulation Of 2-Dimensional Fluidized-Bed". Powder Technology, 77, 79-87.

VAN DEEMTER J. J., (1961). "Mixing and contacting in gas-solid fluidized beds". Chemical Engineering Science, 13, 143-154.

XU B. H., YU A. B., (1997). "Numerical simulation of the gas-solid flow in a fluidized bed by combining discrete particle method with computational fluid dynamics". Chemical Engineering Science, 52, 2785-2809.

YU A. B., XU B. H., (2003). "Particle-scale modelling of gas-solid flow in fluidisation". Journal of Chemical Technology and Biotechnology, 78, 111-121.

ZHANG M. H., CHU K. W., WEI F., YU A. B., (2008). "A CFD-DEM study of the cluster behaviour in riser and downer reactors". Powder Technology, 184, 151-165.

ZHOU Y. C., YU A. B., STEWART R. L., BRIDGWATER J., (2004). "Microdynamic analysis of the particle flow in a cylindrical bladed mixer". Chemical Engineering Science, 59, 1343-1364.

ZHOU Z. Y., KUANG S. B., CHU K. W., YU A. B., (2010). "Discrete particle simulation of particle-fluid flow: model formulations and their applicability". Journal of Fluid Mechanics, 661, 482-510.

ZHU H. P., ZHOU Z. Y., YANG R. Y., YU A. B., (2007). "Discrete particle simulation of particulate systems: Theoretical developments". Chemical Engineering Science, 62, 3378-3396.

ZHU H. P., ZHOU Z. Y., YANG R. Y., YU A. B., (2008). "Discrete particle simulation of particulate systems: A review of major applications and findings". Chemical Engineering Science, 63, 5728-5770.

ZHU J. X., YU Z. Q., JIN Y., GRACE J. R., ISSANGYA A., (1995). "Cocurrent down flow circulating fluidized bed (downer) reactors a state-of-the-art review". Canadian Journal of Chemical Engineering, 73, 662-677.

