Influence of Packing Limit on Gas-Solid Fluidized Bed Flow Pattern by Computational Fluid Dynamics

Kasamawan Taorat Eakarach Bumrungthaichaichan Santi Wattananusorn

Department of Chemical Engineering, Faculty of Engineering,

King Mongkut's Institute of Technology Ladkrabang, Bangkok

Abstract

The present work aims to studythe influence of packing limit on gas-solid fluidized bed flow pattern by using the computational fluid dynamics (CFD). The Multi-fluid Eulerian model was applied to simulate the unsteady stategas-solid fluidized bed. The gas-solid phase was characterized by varying the packing limit values from 0.55 - 0.63. The model was validated by comparingthe simulation results with the experimental data to achieve the optimal model. The simulated resultsshowed that the flow pattern obtained bythe modelwithpacking limit value of 0.63 was the suitable model because of the accurate steady state value and its time dependent pressure drop tendency.

Keywords : Gas-solid fluidized bed, Drag force, Eulerian model, CFD

1. Introduction

Fluidized bed reactors are widely used in modern processing industries which require high mixing rate or segregation phenomena. They are applied to many chemical, petrochemical and food plant operations such as catalytic cracking, combustion, gasification, etc. Fluidized bed which consists of solid particles and pressurized gas is called gas-solid fluidized bed reactor. Although the combined gas and solid phases in the reactor have thorough contact with excellent heat, mass and momentum transfer, their flow behavior is complex making the flow prediction a challenging task. In addition, the efficiency of the fluidized bed reactor depends on several parameters, i.e., gas velocity, particles density, particle size and packing characteristics of fluidized bed. Therefore, the accurate model is required to predict the behavior of multiphase flow in the fluidized bed reactor. Computational fluid dynamics (CFD) is a suitable tool for modeling hydrodynamics of multiphase flow and reducing the number of steps for reactor scaling up and design [1-2].

There are two routes for CFD multiphase flow simulations. The former is called the Euler-Lagrange model, which solves the dispersed phase by tracking a large number of particles through the flow field andusing the assumption that the dispersed second phase occupies a low volume fraction. Meanwhile, the latter is called the Euler-Euler model. It solves the different phase as interpenetrating continua by assuming that the volume of a phase cannot be occupied by other phase. The volume fraction is continuous functions of space and time. Three different Euler-Euler models are available, including, the volume of fluid model (VOF), the mixture model and the Eulerian model [3]. The VOF model is designed for two or more immiscible fluids where the position of the interface between the fluids is of interest. Application for the VOF include stratified flows, free surface flow, filling, the motion of bubbles in a liquid, the motion of liquid after a dam break etc. The mixture model is designed for two or morephase (fluid or particulate) which are treated as interpenetrating continua.

Applications of the mixture model include particle-laden flows with low loading, bubbly flow, sedimentation, cyclone separators etc. The Eulerian model is designed for the complex of the multiphase and solves a set of momentum and continuity equation for each phase. Coupling is achieved through the pressure and interphase exchange coefficients. Application of the Eulerian multiphase model include bubble columns, particle suspension and fluidized beds.[4].

In CFD simulation, the mass and momentum of the gas-solid fluidized beds are calculated by using the Eulerian model and kinetic theory for granular flow [1,3]. The momentumtransfer between gas and solid interphase is represented by using drag force. Further, the drag of particle can be affected by other particles. The momentum exchange coefficient of gas-solid systems are calculated by the Syamlal-O'Brien [1], Gidaspowand Wen and Yu drag fraction [1-2,5].

The objective of this study is to predict the simulation model of the gas-solid fluidized bed by using the CFD technique. The simulated results were compared with the experiment. The simulation models were assumed to betwo dimensional. The FLUENT15 CFD program is based on the finite volume method and Syamlal-O'Brien drag function was applied to calculate momentum exchange coefficients. The effect of packing limit on flow pattern wasstudied to obtainthe accurate model for predicting fluidized bed reactor.

2. Simulation setup

2.1 Geometry model and Gas-solid properties

The geometry of fluidized bed reactor was similar to the experiment of Taghipour et al. [1]. The cylindrical reactor has a 0.4 m height and 0.28 m diameter. The solid beds were spherical glass beads of 250-300 μm diameter (275 μm of mean particle diameter), 2,500 kg/m^3 density and 0.28 mm static bed height. The inlet gas was air with a density of 1.225 kg/m^3 at a velocity of 0.38 cm/s. The 2D schematic of fluidized bed reactor is showed in Figure 1. The fluidized bed model were simulated by varying five different packing limit values (0.55, 0.57, 0.58, 0.60 and 0.63).

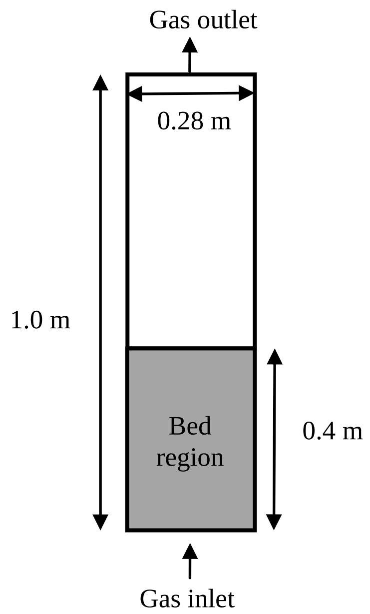


Figure 1. 2D schematic of fluidized bed reactor

2.2 Modeling parameters

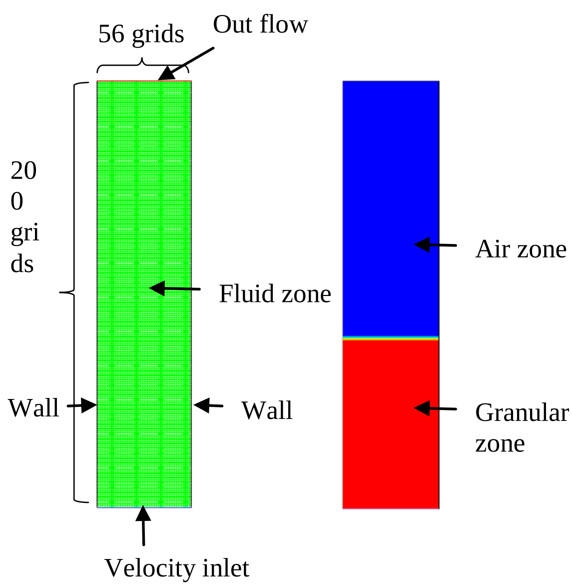


Figure 2. 2D of mesh resolution and boundary conditions

Gambit software was used togenerate the grid of 2D fluidized bed. The 56 x 200 rectangular grids were achieved by 0.005 m of grid interval spacing in width and height reactor. A time step of 0.001 s and 20 iterations per time step were used. Regarding to the previous literature, for the glass beads of 300 µm diameter, the restitution coefficients is 0.9 [6]. The boundary conditions of inlet was velocity inlet while outlet was outflow. Moreover, the no-slip boundary condition was applied at the walls.Figure 2 shows 2D mesh resolution and boundary conditions.

2.3 Computational models

CFD simulation was performed by using Fluent 15 software. The multiphase flow Eulerian model was used for considering the mass and momentum conservation of gas and solid phase. The mass conservation equations are given by

Gas phase:

$$\frac{\partial}{\partial t}\cdot\left(\alpha_g\cdot\rho_g\right)+\nabla\cdot\left(\alpha_g\cdot\rho_g\cdot\vec{v}_g\right)=0 \quad (1)$$

Solid phase:

$$\frac{\partial}{\partial t}\cdot\left(\alpha_s\cdot\rho_s\right)+\nabla\cdot\left(\alpha_s\cdot\rho_s\cdot\vec{v}_s\right)=0 \quad (2)$$

The momentum conservation equations are given by

Gas phase:

$$\frac{\partial}{\partial t}\cdot\left(\alpha_g\cdot\rho_g\cdot\vec{v}_g\right)+\nabla\cdot\left(\alpha_g\cdot\rho_g\cdot\vec{v}^{2}{}_g\right) = -\alpha_g\cdot\nabla p+\nabla\cdot\overline{\overline{\tau}}_g + \alpha_g\cdot\rho_g\cdot\vec{g}+K_{gs}\cdot\left(\vec{v}_g-\vec{v}_s\right) \quad (3)$$

Solid phase:

$$\frac{\partial}{\partial t}\cdot\left(\alpha_s\cdot\rho_s\cdot\vec{v}_s\right)+\nabla\cdot\left(\alpha_s\cdot\rho_s\cdot\vec{v}^{2}{}_s\right) = -\alpha_s\cdot\nabla p+\nabla\cdot\overline{\overline{\tau}}_s + \alpha_s\cdot\rho_s\cdot\vec{g}+K_{gs}\cdot\left(\vec{v}_g-\vec{v}_s\right) \quad (4)$$

The Syamlal-O'Brien drag function is used for considering the gas-solid momentum exchange coefficient, K_{sg}, is given by

$$K_{gs}=\frac{3}{4}\cdot\frac{\alpha_s\cdot\alpha_g\cdot\rho_g}{\nu^{2}{}_{r,s}\cdot d_s}\cdot C_D\cdot\left(\frac{\mathrm{Re}_s}{\nu_{r,s}}\right)\cdot\left|\vec{v}_s-\vec{v}_g\right| \quad (5)$$

Where

$$\nu_{r,s}=0.5\cdot\left(A-0.06\cdot\mathrm{Re}_s+\sqrt{(0.06\cdot\mathrm{Re}_s)^2+0.12\cdot\mathrm{Re}_s\cdot(2\cdot B-A)+A^2}\right) \quad (6)$$

$$A=\alpha_g{}^{4.14}, B=0.8\alpha_g{}^{1.28} \text{ for } \alpha_g\le 0.85$$

$$A=\alpha_g{}^{4.14}, B=\alpha_g{}^{2.65} \text{ for } \alpha_g\ge 0.85$$

and

$$C_D=\left(0.63+\frac{4.8}{\sqrt{\mathrm{Re}_s/\nu_{r,s}}}\right)^2 \quad (7)$$

The constitutive equations and simulation setting of fluent software are shown in Table1.

Table 1.The constitutive equations and simulation setting [7].

Description	Base case setting/value
Operating conditions	
Mesh resolution	56 ×200 grids
Convergence criteria	10^{-3}
Maximum iterations	20
Discretization method	First order upwind
Time step	0.001 s
Reactor width × length	0.28 × 1.0 m^2
Bed particle width × length	0.28 × 0.4 m^2
Restitution coefficient	0.9
Outlet boundary condition	Out flow
Wall boundary condition	No slip shear condition
Gravitational acceleration	9.8 m/s^2
Operating pressure	1.01325 × 10^5 Pa
Gas superficial velocity	0.38 m/s
Inlet boundary condition	Gas velocity inlet
Constitutive equation	
Granular viscosity	Syamlal-obrien
Granular bulk viscosity	Lun-et-al
Frictional viscosity	Schaeffer
Angle of internal friction	30
Frictional Pressure	Based-ktgf
Friction packing limit	0.61
Granular conductivity	Syamlal-obrien
Drag law	Syamlal-obrien
Coefficient of restitution	0.9

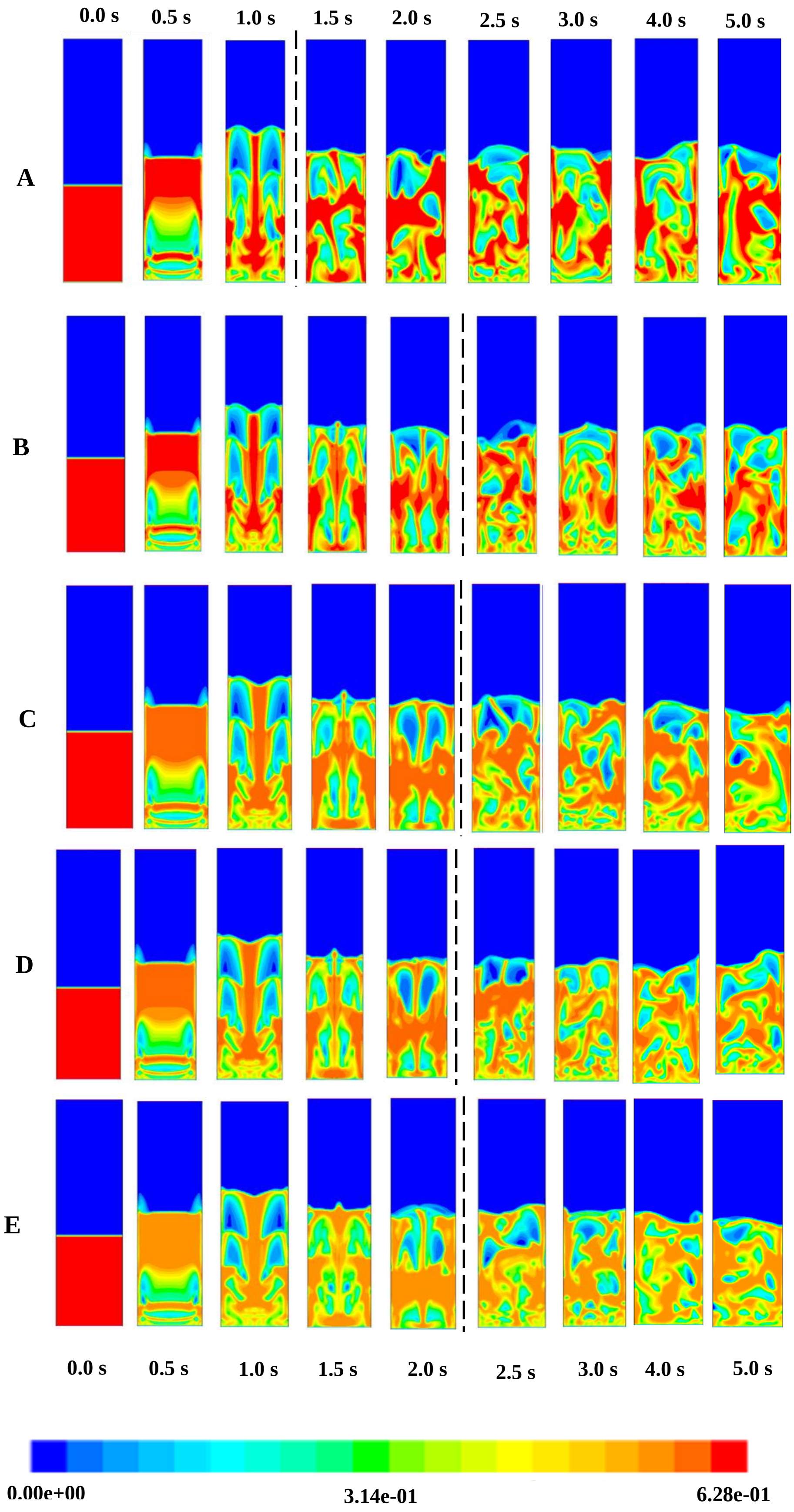


Figure 3. Solid volume fraction contour for packing limit of

(A) =0.63, (B) = 0.60, (C)= 0.58, (D) = 0.57, (E)= 0.55

3. Results and discussion

3.1 Solid volume fraction profile

The contour plot of solid volume fraction for different packing limit values are depicted in Figure 3. At the initial, all of fluidized beds models were impulsively fluidized by inlet gas with a velocity of 0.38 m/s. The increasing of bed height is due tothe bubble formation inside the column. The models exhibit the different steady state values. This value can be identified by considering the unchanged bed height and flow pattern. The results revealed that the steady state prediction obtained by the model with packing limit of 0.63 (1.5 s) is more accurate than other models as compared with the experiment [1].

3.2 Pressure drop

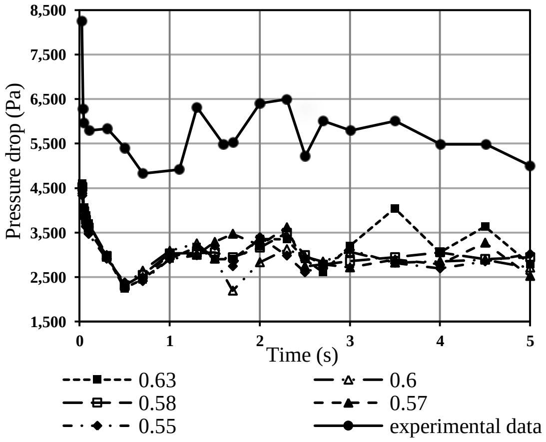
In order to confirm the accuracy of present simulation, the predicted time dependent pressure drop for different models were compared with the experiment [1] as shown in Figure 4. Figure 4 shows that the predicted pressure drop of these models are different and exhibit the under prediction values. However, the tendency of predicted time dependent pressure drop for five different models were investigated by using Eq. (8) to obtain the optimal packing limit value for this simulation.

$$\sum_{i=1}^{N-1}\left(X^*_i\right)=\sum_{i=1}^{N-1}\sqrt{\left[\left(\frac{\Delta\theta_{i,\exp}}{\Delta t_{i,\exp}}\right)-\left(\frac{\Delta\theta_{i,sim}}{\Delta t_{i,sim}}\right)\right]^2} \tag{8}$$

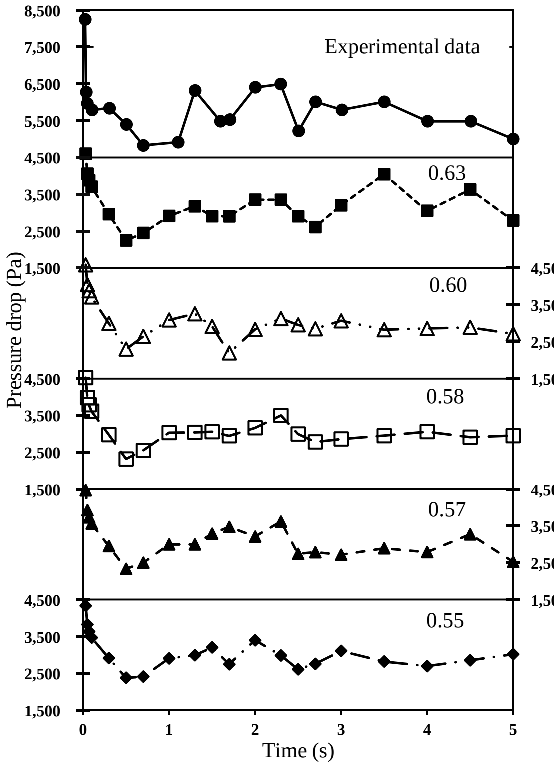
where θ is pressure drop (Pa), t is time (s) and N is number of experimental data.

From Eq. (8), it can be seen that the smaller value of $\sum_{i=1}^{N-1}\left(X^*_i\right)$ results more accurate model. These values of five different models are summarized in Table 2.

From Table 2, the results revealed that the model with packing limit of 0.63 shows the smallest values of $\sum_{i=1}^{N-1}\left(X^*_i\right)$. Hence, it can be deduced that the tendency of time dependent pressure drop obtained by the model with packing limit of 0.63 is more accurate as compared to the other models. In other words, the model with packing limit of 0.63 is the suitable values for predicting the fluid flow inside the fluidized bed reactor.



(a)



(b)

Figure 4. Time dependent of pressure drop
(a) Comparison of pressure drop for different packing limit values
(b) Enlargement of comparison of pressure drop for different packing limit values

Table 2. The variation of flow pattern.

Models	$\sum_{i=1}^{N-1}\left(X^{*}_{i}\right)$[$10^6$ (Pa/s)]
Packing limit 0.63	0.23979
Packing limit 0.60	0.24070
Packing limit 0.58	0.24141
Packing limit 0.57	0.24067
Packing limit 0.55	0.24188

4. Conclusions

The multifluid Eulerian model can be employed to predict the flow inside gas-solid fluidized bed. The simulated results were in good agreement with the experimental data.The prediction accuracy can be improved by using a suitable packing limit value. The model with packing limit value of 0.63 was the most optimal model for predicting the gas-solid fluidized bed reactor because of its steady state value, predicted pressure drop tendency, and probably the drag force generated by the influence of packing limit.

5. Notation

5.1 Symbols

C_D	drag coefficient	[-]
d_i	diameter	[m]
g	acceleration of gravity	[m/s^2]
K_{gs}	gas/solid momentum exchange coefficient	[-]
N	number of experimental data	[-]
p	pressure	[Pa]
r	radial coordinate	[m]
Re	Reynolds number	[-]
t	time	[s]
θ	pressure drop	[Pa]
v_i	velocity	[m/s]
α_i	volume fraction,	[-]
ρ_i	density	[kg/m^3]
$\bar{\bar{\tau}}_i$	stress tensor	[Pa]

5.2 Subscripts

g	gas
i	general index
s	solid
exp.	experimental data
sim	simulation model

7. References

[1] F. Taghipour, N. Fllis, C. Wong, “Experimental and Computational Study of Gas-Solid Fluidized Bed Hydrodynamics,” Chemical Engineering Science,Vol.60, pp.6857-6867, July, 2005.

[2] J. T. Cornelissen, F. Taghipour, R. Escudie, N. Eillis, J.R. Grace, “CFD Modelling of a Liquid-Solid Fluidized Bed,” Chemical Engineering Science, Vol.62, pp.6334-6348, July, 2007.

[3] A. Kumar, CFD Modeling of Gas-Liquid-Solid Fluidized Bed, NationalInstitute of Technology Rourkela, 2008-2009.

[4] Fluent, Inc., Fluent 6.1UDF Manual, 2003.

[5] T. Li, A. Gel, S. Pannala, M. Shahnam, M. Syamlal, “CFD Simulation of Circulating Fluidized Bed Risers, Part I: Grid Study,” Powder Technology, Vol. 254, pp.170-180, January, 2014.

[6] C.C Pain, S. Mansoorzadeh, C.R.E. de Oliveira, “A Study of Bubbling and Slugging Fluidized Beds Using the Two Fluid Granular Temperature Model,” International Journal of Multiphase Flow, Vol.27, pp.527-551, April, 2000.

[7] Ansys, Inc., Ansys Fluent 12.0 Theory Guide, 2009