Gas-solid two phase flow simulation in rectangular riser

Is Bunyamin Suryo1, Tri Yogi Yuwono1, Uwe Schnell2
1 Mechanical Engineering Department, Institut Teknologi Sepuluh Nopember
Kampus Keputih Sukolilo, Surabaya 60111, Indonesia
*Email: isbunyamin@gmail.com

2 Institut für Feuerungs- und Kraftwerkstechnik (IFK), Stuttgart University
Pfaffenwaldring 23, 70569 Stuttgart, Germany

Abstract. The numerical studies of gas-solid two phase flow in a rectangular riser are carried out and evaluated. Eulerian-Eulerian approach using kinetic theory of granular flow is utilised in the simulation. Syamlal et al. drag model is utilised to calculate the energy exchange between gas and solid phase. Radial distribution model of Syamlal et al. and the restitution coefficient of 0.7 are used to predict the hydrodynamics flow in the riser. The simulation is carried out from 0 s to 40 s using time steps of 0.00015 s and on a single computer. The comparisons of numerical results with experimental data are made using lateral profiles of the solid phase velocity and the gas volume fraction. The simulation result overestimates the experimental value due to the coarse nature of the grid and probably due to uncertainties of the inlet flow properties.

Keywords: gas-solid flow, numerical study, eulerian-eulerian approach, drag model.

1. Introduction
The complexity of hydrodynamics in gas-solid flow in column reactors such as the interaction between phases, mixing patterns, and particles distribution induces a complication to the experimental techniques in determining the hydrodynamics phenomena inside the reactor. Furthermore, in order to achieve a detailed analysis of hydrodynamics phenomena using experimental techniques, varying numbers of parameters, such as gas velocity, particle diameter and density, reactor geometry, is mandatory. These variations generate a highly cost-demanding experiment. Moreover, the experimental techniques have limits in capturing complex hydrodynamics phenomena in fluidized beds such as solid particles clusters and distribution.

Regarding those shortcomings of the experimental techniques in predicting the gas-solid hydrodynamics, the numerical simulation method comes to handle it. In the past decades, many researchers have shown that numerical simulation methods have great potential to study gas-solid hydrodynamics [1]. By the development of the method of numerical study, computational fluid dynamics (CFD) became an important method to exercise the gas-solid hydrodynamics. However, the accuracy of CFD models must be validated using experimental data.

In the numerical study of fluid dynamics, generally two different approaches are utilised in simulating gas-solid two phase flows, namely Eulerian-Lagrangian and Eulerian-Eulerian approach.
The difference between these two approaches is how the model treats the solid phase. Generally, the Eulerian-Lagrangian method is utilised for systems with its number of solid particles less than 100,000 particles and instantaneous interaction between particles has to be taken into account [2]. Whereas the Eulerian-Eulerian model is generally used for a system of large size in which there is no need to take into account the details of the hydrodynamics behaviour of gas-solid flow. In this paper, the Eulerian-Eulerian model using the kinetic theory of granular flow is implemented into a CFD code. This code called “AIOLOS” is a computational fluid dynamics software which has been developed by Institut für Feuerungs- und Kraftwerkstechnik (IFK), Stuttgart University, Germany. The developed AIOLOS is then tested to exercise and analyse the hydrodynamics mannerism of gas-solid flow in a 3-dimensional rectangular riser at transient conditions. Afterwards, the time-averaged calculation results are compared to the empirical data which are obtained by experiments.

2. Eulerian-eulerian approach for gas-solid two-phase flow

In order to describe gas-solid two phase flow, a set of governing equations is derived by averaging of the local instantaneous values of the equations of motion of a single particle (solid phase) and the Navier-Stokes equations for fluid motion (gas phase) over a region. The Navier-Stokes equations compose of the mass and momentum conservation equation for both phases which are explained below. Mass conservation of gas phase follows equation 1.

$$\frac{\partial}{\partial t}(\varepsilon_g \rho_g) + \frac{\partial}{\partial x_i}(\varepsilon_g \rho_g U_{gi}) = 0$$

(1)

Mass conservation of solid phase follows equation 2.

$$\frac{\partial}{\partial t}(\varepsilon_m \rho_m) + \frac{\partial}{\partial x_i}(\varepsilon_m \rho_m U_{mi}) = 0$$

(2)

Volume fractions comply with equation 3.

$$\varepsilon_g + \varepsilon_m = 1$$

(3)

Conservation of momentum for gas phase follows equation 4.

$$\frac{\partial}{\partial t}(\varepsilon_g \rho_g U_{gi}) + \frac{\partial}{\partial x_j}(\varepsilon_g \rho_g U_{gi} U_{gj}) = -\varepsilon_g \frac{\partial p_g}{\partial x_i} + \frac{\partial \tau_{gij}}{\partial x_j} - I_{gmi} + \varepsilon_g \rho_g g_i$$

(4)

Conservation of momentum for solid phase is according equation 5.

$$\frac{\partial}{\partial t}(\varepsilon_m \rho_m U_{mi}) + \frac{\partial}{\partial x_j}(\varepsilon_m \rho_m U_{mi} U_{mj}) = -\varepsilon_m \frac{\partial p_m}{\partial x_i} + \frac{\partial \tau_{mij}}{\partial x_j} + I_{gmi} + \varepsilon_m \rho_m g_i$$

(5)

In Equations 1 to Equation 5, $\rho$ is density, $\varepsilon$ is volume fraction, $\tau$ is stress tensor, $U$ is velocity, $P_g$ is gas pressure, $g$ is gravity, $P_m$ is solid pressure, and $I_{gmi}$ is the exchange energy between solid and gas phases. The indices $g$ and $m$ represent the gas and solid phase, respectively. While the indices $i$ and $j$ are both from 1 to 3 which denote the coordinate directions for $x$, $y$, and $z$ directions.

In Eulerian-Eulerian approach for gas-solid flow, the interplay between gas and solid phases is presented by the interphase momentum transfer which is expressed by a exchange coefficient of momentum, $\beta_{gm}$, multiplied by the relative velocity between gas and solid phases follows equation 6.

$$I_{gmi} = \beta_{gm}(U_{gi} - U_{mi})$$

(6)

The value of interphase momentum transfer depends on the volume fraction of the solid phase and the relative velocity of the phases. Due to its dominant and important role in the conservation equation of momentum for both phases, the appropriate model for the drag force
coefficient is a must. In the literature, there are several models which could be employed to calculate the drag force coefficient and all of them are expressed as empirical equations and correlations. Some of them are the drag models proposed by Wen-Yu [3], Gidaspow [4], and Syamlal et al. [5].

The kinetic theory of granular flow is employed to calculate the parameters of solid phase flow: solid viscosity and solid pressure. Basically, the theory relies and follows the kinetic theory of gases by Chapman and Cowling [6]. The motion of the solid particle in the gas-solid multiphase flow could be similar to the motion of the gas molecules. In the dense gases, the gas molecules are basically assumed to be very small relative to the distance between molecules, having random motion and frequently colliding with each other. The theory introduces a term called granular temperature to represent the kinetic energy of the random solid particle motion. Regarding Gidaspow [4], the granular temperature equation is assigned as in equation 7.

$$\frac{3}{2} \frac{\partial}{\partial t} (\varepsilon_m \rho_m \theta_m) + \frac{\partial}{\partial x_i} (\varepsilon_m \rho_m U_m \theta_m) = \left( -P_m \delta_{ij} + \tau_{mij} \right): \frac{\partial U_m}{\partial x_i} - \frac{\partial}{\partial x_i} \left( k \frac{\partial \theta_m}{\partial x_i} \right) - \gamma + \zeta$$

(7)

3. Numerical simulation procedure

The solution algorithm used in the simulation is adopted from Syamlal et al. [7] with some modifications. In the present work, Body Fitted Coordinates with a non-staggered structured grid system are used. In order to calculate diffusive and convective flows at the cell-face, the Central Differencing Scheme is employed into the AIOLOS code. For dealing with the pressure-velocity coupling problem, the Semi-Implicit Method for Pressure Linked Equations is utilised [8]. The Date interpolation scheme is utilised to prevent decoupling of velocities and pressure [9]. The Iterative solver is utilized to solve a coupled linear equation system produced by the discretization procedure. In this study, the Successive Over-Relaxation Red Black iterative solver is employed to calculate the linear equation system. For obtaining the solid volume fraction, the technique implemented by Syamlal et al. called the solid volume fraction correction equation is adopted [7]. This technique is mainly based on the idea to incorporate the effect of solid pressure in calculating the solid volume fraction due to the strong correlation between these two properties.

4. Simulation parameter

The test case for gas-solid multiphase flow simulation in this paper is corresponding to the experiments of J. Zhou et al. [10] and its simulation by Tingwen Li et al. [11]. The case is a 3D-rectangular riser and the detailed simulation parameters are listed in Table 1. The geometry of the simulation test case is shown by Figure 1, while Figure 2 shows the rectangular meshing of the computational domain. The details of boundary conditions (BC) of the simulation, namely inlet BC, outlet BC, wall BC, for gas and solid phases are explained below.

The boundary conditions for both phases in the inlet and its properties are listed below:

- **Gas phase**
  - $\mu_g = 1.82 \times 10^{-5} \text{ kg m}^{-1} \text{ s}^{-1}$
  - $\rho_g = 1.225 \text{ kg m}^{-3}$
  - $\varepsilon_g = 0.9$
  - $U_{g,x} = 5.5 \text{ m s}^{-1}$
Solid phase
\[ d_m = 213 \times 10^{-3} \text{ m} \]
\[ \rho_m = 2640 \text{ kg m}^{-3} \]
\[ \varepsilon_m = 0.1 \]
\[ U_{mx} = 0.1515 \text{ m s}^{-1} \]

Concerning the riser dimension and the uniform inlet gas velocity entering the riser, it shows that the gas flow is turbulent due to its Reynolds number which is \( 5.43 \times 10^4 \). This bed riser is then identified as the turbulent bed regime. Regarding the properties of the solid phase, it can be concluded that the used solid particle is classified as Geldart particle Group B ("sandlike" or bubbly particles with sizes between \( 150 \times 10^{-3} \text{ m} \) and \( 500 \times 10^{-3} \text{ m} \) and density from 1400 to 4000 kg m\(^{-3}\)). The zero gradient at outlet for any property is set up as the boundary condition of the outlet. For both phases, gas and solid phase, a no slip boundary condition is utilized.

5. Simulation result and discussion
The simulation is carried out from 0 s to 40 s using Syamlal et al. drag model, the radial distribution function of Syamlal et al., the restitution coefficient of 0.7, and time step of 0.00015 s. The simulation is computed on a single computer. The simulation result from 30-40 s is then averaged in order to be compared with the experiment by J. Zhou et al. [10]. The comparisons of numerical results with experimental data are made using lateral profiles of the solid phase velocity and of the gas volume fraction.

For the lateral profile, the time-averaged of the solid velocity are examined at 5.13 m above the inlet riser as shown by Figure 3. At this position, two different lateral profiles of the solid velocity at \( z=0.073 \text{ m} \) and 0.1 m are highlighted as shown by Figure 4. For \( z=0.073 \text{ m} \), an asymmetric solid velocity profile is captured, as shown by “sim.1”. While for \( z=0.1 \text{ m} \), the solid velocity profile is presented by “sim.2” and it is an asymmetric solid velocity profile as well. Comparing the simulation results to the empirical data of J. Zhou et al. [10], significant differences are discovered for both positions. The simulation result overestimates the experimental value at both positions, namely “exp.1” and “exp.2”. Furthermore, the simulation is not proof to catch the solid down flow phenomenon up near the wall. In general, a poor accordance between the simulation prediction and the experiment is obtained for the lateral profile of the solid velocity.

| Table 1. Simulation parameters of 3D gas-solid flow in rectangular riser |
|----------------------------------|
| Parameter                        | Value                                |
| Riser geometry (rectangular)     | 0.146 m x 0.146 m x 9 m               |
| Meshing (cells)                  | 146168                               |
| Flow condition                   | transient (unsteady)                 |
| Time step (s)                    | 0.00015                              |
| Time interval of simulation (s)  | 0-40                                 |
| Range of time-averaged results (s)| 30-40                               |
| Drag model                       | Syamlal et al. [7]                   |
| Radial distribution model        | Syamlal et al. [7]                   |
| Restitution coefficient           | 0.7                                  |
| Gas phase viscous                | laminar                              |
From the experiment, the solid mass flux is the only available data at the inlet for the solid phase. Throughout the simulation, if the volume fraction of solid phase of the inlet boundary condition is too low then the solid velocity of the inlet boundary condition becomes higher than the actual solid velocity. This situation leads to a higher solid velocity inside the riser due to a higher solid flow momentum at the inlet. In this study, the volume fraction of solid phase in the inlet boundary condition is set as 0.1, based on the work of Tingwen Li et al. [11]. Increasing the solid volume fraction of the inlet boundary condition may decrease the solid velocity inside the riser such that the simulation prediction can be closer to the experiment. Almuttahar and Taghipour [13] and T Shah et al. [14] noted that the flow pattern in the riser is influenced by the feed geometry and flow conditions through the inlet cross section. The drag model and the wall
boundary condition could also contribute to the poor prediction of the calculation compared to the experiment of J. Zhou et al. [10]. The correct description of the inlet boundary condition tends to be critical in successfully predicting the behaviour of flow hydrodynamics in the riser.

Figure 3. The highlighted position for capturing the solid phase velocity

Figure 4. Velocity time-averaged lateral profiles of the solid phase at z=0.073 m and 0.1 m as shown by Figure 3

For the time-averaged, the lateral profiles comparison of the volume fraction of gas phase between the simulation and the experiment is carried out at position of z=0.073 m as shown by Figure 5. At this position, two different lateral profiles of the volume fraction of gas phase at x=7.06 m and x=8.98 m above the riser inlet are examined as shown by Figure 6. For x=7.06 m and x=8.98 m, an asymmetric gas volume fraction profile is captured, as shown by “sim.1” and “sim.2”, respectively. Comparing the simulation predictions to the experimental data of J. Zhou et al. [10], the current simulation has a poor agreement with the experimental result at both positions, “exp.1” and “exp.2”. The simulation shows a dense area with a low gas volume fraction near to one side of the riser walls. The lateral profiles of the gas volume fraction obtained by the simulations do not show a symmetrical profile. Furthermore, an asymmetrical profile and generally an overestimated result are obtained by the simulation for both positions. The reason of the poor agreement between the simulation and the experiment of J. Zhou et al. [10] can be the coarse nature of the grid and/or the flow properties of the inlet boundary condition as explained above.
Figure 5. The highlighted position for capturing the gas volume fraction

Figure 6. Volume fraction time-averaged lateral profiles of the gas phase at x=7.06 m and 8.98 m as shown by Figure 5

6. Conclusion
The comparisons of numerical results with experimental data are made using lateral profiles of the solid phase velocity and of the gas volume fraction. The simulation result overestimates the experimental value due to the coarse nature of the grid and probably due to uncertainties of the inlet flow properties. The correct description of the inlet boundary condition tends to be critical in successfully predicting the behaviour of flow hydrodynamics in the riser. The drag model and the wall boundary condition could also contribute to the poor prediction of the simulation compared to the experiment.

7. References
[1] Gidaspow D, Jung J, and Singh R K, 2004 Powder Technology 148(2) 123-141
[2] Weber M W and Hrenya C M, 2006 Chemical Engineering Science 61(14) 4511-4527
[3] Wen C Y, 1966 Mechanics of fluidization In Chem. Eng. Prog. Symp. Ser. volume 6, pages 100-101
[4] Gidaspow D, 1994 Multiphase flow and fluidization: continuum and kinetic theory descriptions (Academic press)
[5] Syamlal M and O’Brien T J, 1989 Computer simulation of bubbles in a fluidized bed In AIChE Symp. Ser., volume 85, pages 22-31
[6] Chapman S and Cowling T G, 1970 The mathematical theory of non-uniform gases: an account of the kinetic theory of viscosity, thermal conduction and diffusion in gases (Cambridge university press)
[7] Syamlal M, Rogers W, O’Brien T J, 1993 MFIX documentation: Theory guide (USA: National Energy Technology Laboratory, Department of Energy)
[8] Patankar S, 1980 Numerical Heat Transfer and Fluid Flow (New York: Hemisphere publishing corporation)
[9] Date A, 1996 Numerical Heat Transfer 29(4) 441-458.
[10] Zhou J, Grace J R, Lim C J, Brereton C M H, 1995 Chemical Engineering Science 50(2) 237–244
[11] Li T, Pannala S, Guenther C, 2011 Numerical simulations of a circulating fluidized bed with a square cross-section 10th International Conference on Circulating Fluidized Beds and Fluidization Technology
[12] Li T, Gel A, Pannala S, Shahnam M, Syamlal M, 2014 Powder Technology 254 170-180
[13] Almuttahar A and Taghipour F, 2008 Powder Technology 185(1) 11-23
[14] Shah M, Utikar R, Evans G, Tade M, Pareek V, 2011 Industrial & Engineering Chemistry Research 51 1721-1728