Proceeding Series of the Brazilian Society of Applied and Computational Mathematics, Vol. 5, N. 1, 2017.

Trabalho apresentado no CNMAC, Gramado - RS, 2016.

Proceeding Series of the Brazilian Society of Computational and Applied Mathematics

CFD-DEM simulations in a gas-solid fluidized bed

Shyam S. Das[1], Yuri D. Sobral[2]

Departamento de Matemática, Instituto de Ciências Exatas, Universidade de Brasília, Campus Universitário Darcy Ribeiro, 70910-900, DF, Brazil

Francisco R. Cunha[3]

Departamento de Engenharia Mecânica, Faculdade de Tecnologia, Universidade de Brasília, Campus Universitário Darcy Ribeiro, 70910-900, DF, Brazil

Abstract

A gas-solid fluidized bed consists of mixture of gas-solid in which the particles were suspended by an imposed upward flow. In the present paper, we have carried out two dimensional numerical simulations of gas-solid fluidized bed. CFD-DEM (Computational fluid dynamics-discrete element modeling) approach is used to model two phase flow composed of solid particles and gas inside the fluidized bed. It uses Eulerian and Lagrangian methods to solve fluid and solid particles respectively. Gidaspow drag model is used to model the interaction between gas and solid particles. Numerical simulations were carried out for various inlet gas velocities (75 cm/s to 82 cm/s) for the fluidized bed. The aim of the present work to search for linear waves. It is interesting to note that, the amplitude of disturbances increase with the height. Further, it is observed that growth rate of the amplitude of disturbances is linear for $v_g = 75$ cm/s and nonlinear for $v_g = 78$ to 82 cm/s.

Keyword: *Fluidized bed, Computational fluid dynamics, discrete element modeling,*

1 Introduction

Gas-solid fluidized beds are widely used in chemical, petrochemical, metallurgical, environmental and energy industries in large scale operations such as coating, granulation, drying. One of the major advantage of fluidized beds lie in good gas-solid mixing, results in uniform temperature distribution and high rate of heat transfer between bed and an immersed heating surface in the bed. A gas-solid fluidized bed consists of mixture of gas-solid in which the particles were suspended by an imposed upward flow. Fluidization occurs when a gas is pushed upwards through a bed of particles. At a low superficial gas velocity, the drag on each particle is also low, and thus bed height remains unchanged. As the superficial gas velocity increases in the reactor, the pressure drop across the bed also increases. Then, at a certain gas velocity, the upward drag force equals to weight of the bed and the particles are

[1]das.shyamsumanta@gmail.com
[2]yurisobral@gmail.com
[3]frcunha2@gmail.com

free to move. Then, the bed is said to be fluidized. Lack of understating of the fundamental of dense gas-particulate flows in general has lead to severe difficulties in design and scaling up the reactor in industrial scale. Also, in most of cases, the design and scale up of fluidized bed reactors is fully an empirical process based on the primitive studies in pilot scale models which is a time consuming process.

In this scenario, computational fluid dynamics (CFD) will be a very useful tool in understanding the complex physical process associated with the fluidized bed reactors. With the increase in the computer technology over the years, it is now possible to simulate the complex dense particulate flows. Generally, there are two different kinds of approaches to model the gas-solid flows (1) Eulerian-Eulerian (EE) (2) Eulerian-Lagrangian (EL) approach. In Eulerian-Eulerian approach, the gas and the solid are treated as interpenetrating continua. Continuity and momentum equations are written for each phase, and interphase interaction is accounted for through appropriate sources and sinks in the phase momentum equations. This approach requires a constitutive equation for the solid phase to relate the solids stress tensor to the velocity field; the fluid phase is typically modeled as Newtonian. The interphase interaction terms typically involve empirical relationships for drag, heat transfer and other exchanges. In Eulerian-Lagrangian approach, the fluid flow is solved using the continuum equations, and the particulate phase flow is described by tracking the motion of individual particles [6]. Discrete particle models (DPM) have been used for a wide range of applications [6].

The study of stability in a fluidized bed aims to identify the hydrodynamic mechanisms that cause the appearance of the bubbles in these flows. Bubbles are large regions void of particles that travel upstream and alter significantly the pattern of the flow. Therefore, it is extremely important to identify the mechanism behinds bubble formation in order to be able to better understand and control such flows. Over the years several researchers carried out stability analysis for the study of stability in a fluidized bed ([1], [2], [11]). In [1], the authors used the local averaged equations for momentum and continuity equations. The averaged equations are perturbed with small disturbances from the homogeneous fluidization state, and linearized with respect to the perturbations. The author carried out a stability analysis and observed that the particle pressure term has a stabilizing effect and that the particle viscosity acts as a short wave filter. The authors studied the stability of stratified particulate flows to transverse disturbances, leading to gravitational overturning [2]. They have used a one-fluid model in which particles are responsible for the stratification of the flow but do not slip relative to the fluid and do not diffuse. A linear stability analysis and a numerical simulation of the governing equations were performed in order to determine and characterize the instability of the flow. They observed that stratified flows are unstable to transverse disturbances and that the instability is driven by a tilt-and-slide mechanism that creates ascending regions of low concentration of particles and descending regions of high concentration of particles. This mechanism might be related to the formation of bubbles in fluidized beds.

Linear stability analysis was carried out by [5] to evaluate the behavior of concentration waves in polarized fluidized beds. They proposed a non linear model for the fluid-particle interaction force, so that the inertial effects arising from wakes behind the particles are incorporated. They observed that, the growth rate instabilities are considerably reduced when a magnetic field is applied in a fluidized bed of magnetic particles, this effect being most efficient. When the applied field is perpendicular to the flow, there is only a small reduction of growth rates. They also observed that, wake interaction mechanism and the particle pressure are stabilizing mechanisms and that the magnetic field intensities necessary for full stabilization when these terms were not included in the model are greater than those compared to predictions made with these terms in the model.

The aim of the present work is to look for the linear waves in the fluidized bed. For this, we have carried out two dimensional CFD-DEM on numerical simulation in a gas-solid fluidized bed using MFIX code. Simulations were performed at different gas velocities. In order to characterize the wave, we studied the amplitude of the wave at different inlet gas velocities.

2 Mathematical Model

In the present study, fluid motion is solved using volume averaged equations [4] and particle motion is modeled by using spring-dashpot model [10].

2.1 Governing equations

2.1.1 Particle Motion

The equation for both translational and rotational motion for the i^{th} particle are given by

$$\begin{aligned} \frac{d\vec{x_i}}{dt} &= \vec{v_i} && (1)\\ m_i\frac{d\vec{v_i}}{dt} &= \overrightarrow{F_{i,j}^{(c)}} + m_i\vec{g} && (2)\\ I_i\frac{d\vec{\omega_i}}{dt} &= \sum_{\substack{i,j=1\\ i\neq j}}^{N} \overrightarrow{\tau_{i,j}} && (3)\end{aligned}$$

The mass, position vector and velocity vector for the i^{th} particle is $m_i, \vec{x_i}, \vec{v_i}$ and $\vec{g}$ is the acceleration due to gravity.

$$\overrightarrow{F_{i,j}^{(c)}} = \sum_{\substack{i,j=1\\ i\neq j}}^{N} \overrightarrow{F_{i,j}^{(n)}} + \overrightarrow{F_{i,j}^{(\tau)}} \qquad (4)$$

$$\begin{aligned} \overrightarrow{F_{i,j}^{(n)}} &= -k\overrightarrow{\delta_{ij}^{(n)}} - \eta(\vec{v}_{rij}\cdot\vec{n}_{ij})\vec{n}_{ij} && (5)\\ \vec{v}_{rij} &= \vec{v}_i - \vec{v}_j && (6)\\ \overrightarrow{F_{i,j}^{(\tau)}} &= -k\overrightarrow{\delta_{ij}^{(\tau)}} - \eta\vec{v}_{rij} && (7)\end{aligned}$$

The contact force, normal and tangential component of the contact forces between i^{th} and j^{th} particle are $\overrightarrow{F_{i,j}^{(c)}}, \overrightarrow{F_{i,j}^{(n)}}, \overrightarrow{F_{i,j}^{(\tau)}}$. $I_i, \vec{\omega_i}$ and $\overrightarrow{\tau_{i,j}}$ are the moment of inertia, angular velocity and torque acting on the i^{th} particle. $\overrightarrow{\delta_{ij}^{(n)}}, \overrightarrow{\delta_{ij}^{(\tau)}}$ are the normal and tangential overlapping distance between particle i^{th} and j^{th} particles, respectively, k and η are the stiffness constant for the spring and viscous damping coefficent. $\vec{v}_{rij}$ is the relative velocity between i^{th} and j^{th} particle and $\vec{n}_{ij}$ is the unit normal vector along $\vec{v}_{rij}$.

2.1.2 Gas Phase

$$\frac{\partial(\epsilon_g\rho_g)}{\partial t} + \frac{\partial}{\partial x_j}(\epsilon_g\rho_g u_j) = 0 \tag{8}$$

$$\frac{\partial}{\partial t}(\epsilon_g\rho_g u_i) + \frac{\partial}{\partial x_j}(\epsilon_g\rho_g u_j u_i) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu_g\left(\frac{\partial u_i}{\partial x_j}\right)\right] + f_{gp} + \epsilon_g\rho_g g \tag{9}$$

The local mean void fraction is given by ϵ_g. u_i is the local mean gas velocities, ρ_g is the gas density, p is the local mean gas pressure, f_{gp} is the force of interaction between gas and solid particles (drag force). Various models of gas-particle interactions were proposed by researchers. The volumetric gas particle interaction is given by

$$f_{gp} = \beta(v_p - u_i) \tag{10}$$

Where v_p is the particle velocity and u_i is the volume averaged gas velocity, β is the fluid particle interaction coefficient. In the present work, Gidaspow drag model is used for the modeling the interaction between gas and the particles phase.

$$\beta = \begin{cases} \frac{1-\epsilon_g}{d_p^2\epsilon_g}\left(150\frac{1-\epsilon_g}{d_p} + 1.75\rho_g\epsilon_g|\vec{v}_p - \vec{u}|\right), & \epsilon_g \leq 0.8, \\ \frac{3}{4}C_D\frac{|\vec{v}_p-\vec{u}|\rho_g(1-\epsilon_g)}{d_p}\epsilon_g^{-27}, & \epsilon_g > 0.8. \end{cases} \tag{11}$$

$$C_D = \begin{cases} \frac{24\left(1+0.15Re^{0.687}\right)}{Re}, & Re < 1000, \\ 0.48, & Re > 1000. \end{cases} \tag{12}$$

$$Re = \frac{|\vec{v}_p - \vec{u}|\rho_g\epsilon_g d_p}{\mu} \tag{13}$$

The particle diameter is given by d_p, μ is the dynamic viscosity, $\vec{v}_p$ is the particle velocity in the averaged cell. The fluid motion was solved simultaneously with the motion of the particles. Initially, the equations for gas motion are solved. Then, the drag force for each particle is calculated using local gas velocity and particle velocity. Two dimensional CFD-DEM simulations were performed for a fluidized bed using open source MFIX-DEM code [8]. Hexahedral cell elements were used for meshing the geometry. Finite volume method (FVM) was used in code to discretize the governing equations [9]. MUSCL (Monotonic Upstream-Centered Scheme for Conservation Laws) scheme [7] was used to model the convective terms in the momentum equations. The convergence criterion was set at 10^{-4} for all the dependent variables. SIMPLE algorithm [9] was used for pressure-velocity coupling. BICGTAB (Biconjugate gradient method) was considered for solving the linear equations. First order Euler integration method was used for time discretization. It was earlier reported from the work of Tsuji et al. [6] that, the time step size in DES simulation depends on the stiffness. Therefore, in the present simulation, $\Delta t = 0.0002s$ is considered. For each time step, convergence criteria was set at 10^{-4} for all the dependent variables. Numerical simulations were performed for $10s$. The inlet boundary of the domain was defined as the velocity inlet. Whereas, at the outlet, pressure outlet boundary condition was used. No slip boundary condition was used at the wall.

3 Results and discussion

In the present section, the results obtained from numerical simulations from a two dimensional fluidized bed was presented in the velocity range $75cm/s$ to $82cm/s$. The simulation parameters were shown in Table 1.

Table 1: Simulation parameters for fluidized bed simulations

Bed dimension ($W \times H$)	$2cm \times 40cm$
Fluid mesh size	2×40
Cell dimensions	$1cm \times 1cm$
Gas velocity at the inlet: (u_i)	75 to $82\frac{cm}{s}$
Gas density(ρ_g)	$1.205E^{-3}\frac{g}{cm^3}$
Gas viscosity(μ_g)	$1.800E^{-4}$
Number of particles(N)	2000
Particle density(ρ_p)	$1.5\frac{gm}{cm^3}$
Particle diameter(d_p)	$1mm$
Particle stiffness coefficient(k)	$1e^{06}$
Particle coefficient of restitution(e)	0.9
Friction damping coeff.(η)	0.1

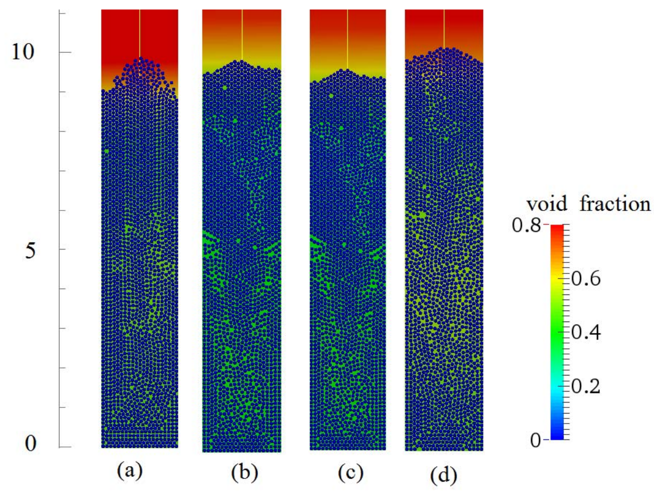

Figure 1: Void fraction plot for different v_g (a) 75 cm/s (b) 78 cm/s (c) 80 cm/s (d) 82 cm/s

Initially, the minimization fluidization velocity (U_{mf}) for the reactor was computed from the simulation. In the present case, U_{mf} is found to be 74.5 cm/s. Fluidization in the reactor began when the inlet gas velocity is equal to the minimization gas velocity. It means, particles in the bed start moving slowly. Figure 1 shows the snapshot for the void fraction

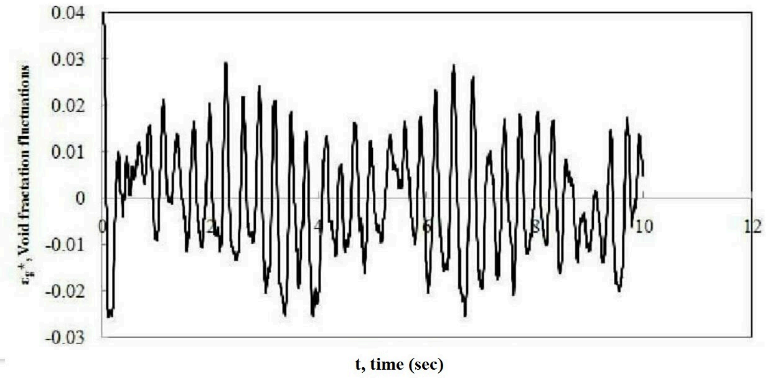

Figure 2: Gas volume fraction fluctuations profile for v_g = 75 cm/s.

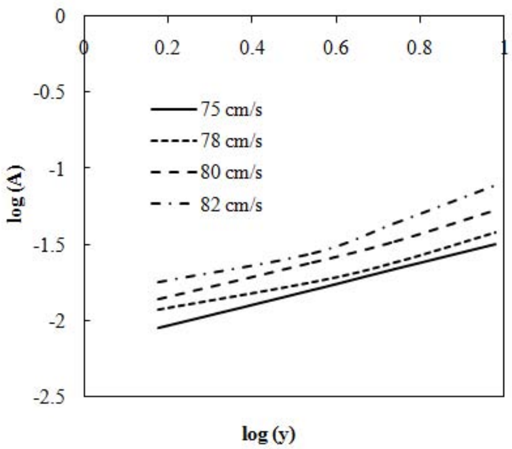

Figure 3: Amplitude curve for different inlet gas velocities.

plot for different inlet gas velocities. The void fraction plots show that with the increase inlet gas velocities, particle motion increases in the fluidized bed. Figure 2 shows the plots of gas volume fraction fluctuations ($\varepsilon_g^* = \varepsilon_g - \varepsilon_{gavg}$) for v_g = 75 cm/s at the lower part of the bed (y = 1.5 cm). It is clearly observed that, the presence of periodic motion inside the fluidized bed. Figure 3 shows the variation of the amplitude of the wave with height of the bed for different inlet gas velocities. Near the lower part of the bed, the amplitude of the disturbance is small. As the height increases, the amplitude of the wave also increases. Further, it is observed from the simulation that as the inlet gas velocity increases, amplitude of these waves also increase. The increase in amplitude of the wave is attributed due to the supply of more energy at the inlet of fluidized bed. It is interesting to note that, these waves grow linearly in the velocity range v_g = 75 cm/s. For v_g = 78 to 82 cm/s, the growth of the waves are non linear (Figure 3).

DOI: 10.5540/03.2017.005.01.0265

4 Conclusion

Two dimensional numerical simulations were performed for the different inlet gas velocities in a fluidized bed. CFD-DEM approach was used to model the gas-solid flows. From the simulation, we were able to capture the presence of linear waves form for $v_g = 75$ cm/s. It was further observed that, near the lower part of the bed, the amplitude of the disturbance is small. As the height increases, the amplitude of the wave also increases. Further, it is found that as the inlet gas velocity increases, amplitude of these waves also increases. The amplitude of these waves grows linearly for the velocity $v_g = 75$ cm/s. It is interesting to note that, non linear growth of the amplitude of the disturbances is observed for the velocity range $v_g = 78$ cm/s to 82 cm/s. In our future work, we have planned to carry out stability analysis with different particle size, bed dimensions.

References

[1] Y.D. Sobral, F.R. Cunha. A linear stability analysis of a homogeneous fluidized bed. *Tendencies in Computational and Applied Mathematics* 3 (2) (2002) 197-206.

[2] Y.D. Sobral, E.J. Hinch, Gravitational overturning in stratified particulate flows. *SIAM Journal on Applied Mathematics.*, 71 (2011) 2151-2167.

[3] Y.A. Sergeev, Linear and Non-linear Concentration Waves in Magnetically Stabilized Fluidized Beds, *Mobile Particulate Systems, NATO ASI Series (1994).* 287,1995, pp. 249-260.

[4] T.B. Anderson, R. Jackson, A fluid mechanical description of fluidized beds: Equations of motion, *I&EC Fundamentals*, 6, No. 4 (1967), 527-539.

[5] Cunha, F. R., Sobral, Y. D., Gontijo, R. G. 2013, Stabilization of concentration waves in fluidized beds of magnetic particles. *Powder Technology*, v. 24, p. 219-229.

[6] Y. Tsuji , T. Kawaguchi , and T. Tanaka, Discrete particle simulation oftwo dimensional fluidized bed, *Powder Technology*, 77 (1993), 79-87.

[7] B. van Leer , Towards the Ultimate Conservative Difference Scheme, V. A Second Order Sequel to Godunov's Method, *J. Com. Phys.* (1979), 32, 101-136.

[8] M. Syamlal, Mfix documentation: Numerical guide. *Tech. Rep. DOE/MC31346-5824, NTIS/DE98002029, (1998) National Energy Technology Laboratory*, Department of Energy, http://www.mfix.org

[9] S. Patankar, Numerical Heat Transfer and Fluid Flow. *Hemisphere Publishing Corporation.*, (1980).

[10] Cundall, P.A., Strack, O.D.L. ,A discrete numerical model for granular assemblies, *Geotechnique* 29 , 47-65, (1979).

[11] Sobral, Y. D., Instabilities in fluidized beds, doctoral thesis, *Department of Applied Mathematics and Theoretical Physics, University of Cambridge.*, England(2008).

