Paper presented in the National Seminar on Recent Advances in Chemical Engineering, 30th - 31st January 2010 (RACE-2010), Department of Chemical Engineering, Gandhi Institute of Engineering and Technology, Gunupur, Orissa, India-765022.

CFD SIMULATION OF BED VOIDAGE BEHAVIOUR OF A LIQUID-SOLID FLUIDIZED BED

H. M. JENA[1], RAHUL NAIR[2] and A. CHIDAMBARAM[3]

Department of Chemical Engineering, NIT, Rourkela-769008, INDIA

Email: [1] hara.jena@gmail.com, [2]rahulnair1402@gmail.com, [3] a.chidambaram09@gmail.com

Abstract:
A two dimensional transient model has been developed to simulate the hydrodynamic behaviour of the liquid–solid fluidized bed using the computational fluid dynamics (CFD) method. Eulerian Granular Multiphase model has been used in the present study and simulations have been carried out using the commercial CFD package Fluent 6.2.16. To validate the results of CFD simulation, experiments have been conducted in a 0.1m ID, 1.24m height vertical Plexiglas column. Water and glass beads have been used as the liquid and solid phase respectively. The simulation results are in good agreement with the experimental values. Bed voidage values in the expanded fluidized bed obtained from CFD simulation has been compared with the ones predicted from the equation available in literature.

***Keywords*: liquid-solid fluidization, cfd, pressure drop, bed expansion.**

1. INTRODUCTION

Over the past years fluidization has attracted more and more the attention of the industries. This new technique achieved soon widespread use in several industrial applications particularly in fields of catalytic cracking and coal gasification. Fluidization has always lived up to the expectations, turning into a well established technology which is nowadays employed in many areas such as coal combustion, biomass gasification and waste water treatment. Although extensively used, fluidized beds have always been hard to design, since their performance is highly dependent on the suspension fluid dynamics. The change is significant and is strongly affected by wide range of parameters such as the fluid velocity, density and viscosity the particle size, shape and density, bed configuration etc. [1].

Fluidization basically refers to the process of passing a fluid upwards through a packed bed of solid particles resulting in a pressure drop due to the drag force of the fluid. If the fluid velocity is gradually increased then the pressure drop increases as well as the drag force on the particles and ultimately after some time the particles will no longer be in a state of rest but will start to move and will remain suspended in the fluid. This condition represents fluidization. This suspension has behaviour similar to the dense fluid. This fluidity is one of the main advantages of the use of fluidization for handling solids. If the velocity of fluid is allowed to increase further it is noticed that the pressure drop becomes constant but the bed height continues to increase. The superficial velocity at this point is called the minimum fluidization velocity (U_{mf}). An equation for minimum fluidization velocity can be obtained by setting the pressure drop across the bed equal to the weight of the bed per unit area of cross section, allowing for the buoyant force of displaced liquid. Thus, an overall view on fluidization can be described as when a liquid or a gas is passed at very low velocity up through a bed of solid particles, the particles do not move. The pressure drop can be calculated by the Ergun equation [2].

$$\frac{\Delta P}{H}=150\frac{(1-\varepsilon)^2}{\varepsilon^3}\frac{\mu U}{(\varphi d_p)^2}+1.75\frac{1-\varepsilon}{\varepsilon^3}\frac{\rho U^2}{\varphi d_p} \quad (1)$$

Liquid-solid fluidization refers to holding solid particles in suspension by a liquid. The principles of liquid-solid fluidization have been applied in many areas of engineering applications [3]. Liquid fluidization was first applied nineteenth century for both sorting and sizing of particles in the mineral dressing industry that time it was called as teeter condition. It was only after the term 'fluidization' was coined to describe mobile swarms of particles contacted upwardly by gases that the same term began to be used for analogous liquid-solid contacting, and the term 'teeter bed' was replaced by 'liquid-fluidized bed'. The liquid-fluidized beds are used in processes like; particle classification, backwashing of downflow granular filters, in situ fluidized washing of soils, seeded crystal growth, leaching, adsorption and ion exchange, flocculation in order to achieve clarification of turbid liquids, electrolysis, fluidized bed electrode applications, liquid-fluidized bed heat exchange, thermal energy storage, fluidized bed bioreactors etc. [4].

In two-phase (liquid-solid) upward systems, solid particles whose density is larger than that of the liquid are fluidized by upward liquid. The successful design and operation of such a system depends on the prediction of the fundamental characteristics. The sizing of such a system entirely depends upon the expansion characteristics of the solid phase with the variation in the flow of different fluid phases. The complex hydrodynamics of these reactors are not well understood due to complicated phenomena such as particle–particle, liquid–particle interactions and thus on the bed expansion characteristics. For this reason, computational fluid dynamics (CFD) has been promoted as a useful tool for understanding multiphase reactors for precise design and scale up [5].

The most fundamental consideration in CFD is how one treats a continuous fluid in a discretized fashion on a computer. Computational Fluid Dynamics (CFD) is a valuable research means, for it allows simulating and investigating directly the behavior of full size systems without the need to draw on uncertain results obtained from empirical relations. One of the main advantages of CFD is that there is no need of accurate models to be drawn along with appropriate constitutive equations. CFD is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. CFD is a valuable tool to study the dynamics of things that flow. Using CFD, a computational model can be build that represents a system or device under study. After the required design is built, the fluid flow physics and chemistry to this virtual model is applied and correspondingly the software outputs a prediction of fluid dynamics and related physical phenomena [6].

From a modeling perspective, Roy and Dudukovic [7] have investigated liquid–solid fluid dynamics in a circulating fluidised bed riser using non-invasive flow methods and discussed the solid flow structure in the riser. They have developed a CFD model for the riser and validated the findings with experimental data. Cheng and Zhu [8] have developed a CFD model for liquid–solid circulating fluidised bed reactor and included turbulence and kinetic theory of granular flow in the governing equations to model the high Reynolds number two phase flows with strong particle–particle interactions. They have found enhanced non-uniformities in flow structure for the larger particle system. Doroodchi et al. [9] have used CFD to investigate the influence of inclined plates on the expansion behaviour of solids in a liquid fluidised bed containing two different sized particles. The authors have modelled the drag between the particles and continuous fluid based on experimentally determined Richardson–Zaki exponents for the various particle sizes. Paneerselavam et al. [10] have carried out CFD based investigations on hydrodynamics and energy dissipation due to solid motion in liquid fluidised bed. The CFD simulation of a simple liquid-solid fluidized bed analysing the bed dynamics like bed expansion and minimum fluidization velocity is scarce.

In the present work the hydrodynamic parameter bed voidage (bed expansion) has been studied. The effect of the vaiables viz. initial static bed height, particle size and liquid velocity has been studied on the hydrodynamics. A two dimensional transient model has been developed to simulate the hydrodynamic behaviour of the liquid–solid fluidized bed using the computational fluid dynamics (CFD) method. Eulerian Granular Multiphase model has been used in the present study and simulations have been carried out using the commercial CFD package Fluent 6.2.16. To validate the results of CFD simulation experiments have been conducted in a 0.1m ID, 1.24m height vertical Plexiglas column. Water and glass beads are used as the liquid and solid phase respectively.

2. Computational flow model

An Eulerian granular multiphase model has been adopted where liquid and solid phases are all treated as continua, inter-penetrating and interacting with each other everywhere in the computational domain. The pressure field is assumed to be shared by all the three phases, in proportion to their volume fraction. The motion of each phase is governed by respective mass and momentum conservation equations. For the calculation of liquid–solid inter phase drag force Gidaspow drag model [11] has been used. For solid pressure calculation model based on the kinetic theory of granular flow (KTGF) has been used. Laminar viscous model has been used along with Eulerian Granular multiphase model.

3. Numerical Methodology

The model equations have been solved using the commercial CFD software package Fluent 6.2.16. The fluidised bed considered for the present simulation study is a cylindrical Plexiglas column of diameter 0.1m and height 1.24m. Fig. 1(a) depicts the typical numerical mesh used for this simulation. The governing equations are discretized using element based finite volume method and for spatial discretization of the governing equations, high-resolution discretization scheme is applied which accounts for accuracy and stability. For time discretization of the governing equations, a first order backward Euler scheme is used. The discretized equations are solved using the advanced algebraic multi-grid solver (AMG) technology of Fluent 6.2.16. Fig. 1(b) shows the general procedure for the simulation using Fluent software. Two dimensional computational geometry of the fluidization column have been generated by using top-down technique (Face primitive - Rectangle) by using commercial software GAMBIT 2.2.30 as shown in Fig 1(a).

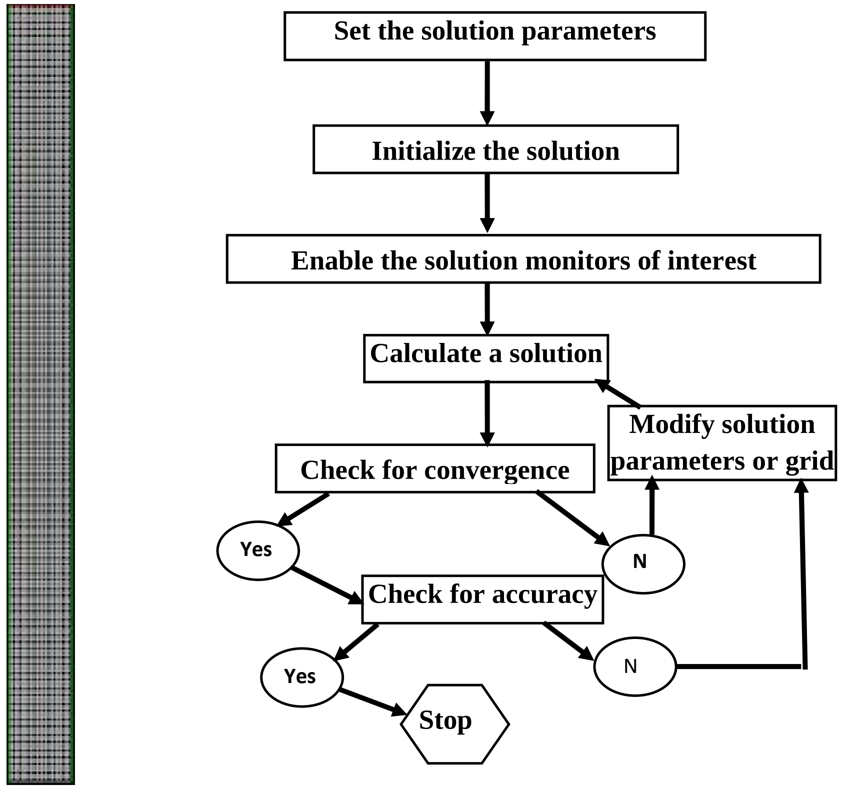


Fig. 1(a) **Fig. 1(b)**

Fig. 1. (a) 2D mesh, (b) Flowchart showing the general procedure for the simulation using Fluent.

After geometry creation, a uniform mesh has been generated with map structured quadrilateral elements containing height to width ratio of 1. Totally 4960 cells with size of each cell 0.005m x 0.005m have been used for computation.

In order to obtain a well-posed system of equations, reasonable boundary conditions for the computational domain have to be implemented. Inlet boundary condition is a uniform liquid velocity at the inlet, and outlet boundary condition is the pressure boundary condition, which is set as1.013×10^5 Pa. Wall boundary conditions are no-slip boundary conditions for the liquid phase and free slip boundary conditions for the solid phase. For higher viscous effect and higher velocity gradient near the wall have been dealt with the standard wall function method. At initial condition the solid volume fraction of 0.59 up to the static bed height of column has been used and the volume fraction of the gas at the inlet and in the free board region is based on the inventory. The process conditions used for simulations are shown in Table 1.

Table 1. Simulation process conditions

Particle size	1.55 mm, 2.18 mm
Particle density	2470 kg/m^3
Initial static bed height	0.171 m
Initial solid holdup	0.59
Superficial liquid velocity	0.004246-0.1019 m/s

The solution procedure involves the following steps: (i) generation of suitable grid system; (ii) conversion of governing equation into algebraic equations; (iii) selection of discretization schemes; (iv) formulation of the discretized equation at every grid location; (v) formulation of pressure equation; (vi) development of a suitable iteration scheme for obtaining a final solution. The Phase Coupled SIMPLE method has been chosen for pressure–velocity coupling. The first-order upwind scheme has been used for discretization of momentum, and volume-fraction equations. The time step size of 0.001s has been used. The convergence criteria for all the numerical simulations are based on monitoring the mass flow residual and the value of $1.0e^{-03}$ was set as converged value. The residual plot of the progress of the simulation is shown in Fig. 2.

The following under relaxation factors have been used for different flow quantities: pressure = 0.3, density = 1, body forces = 1, momentum = 0.2, volume fraction = 0.5, granular temperature = 0.2. The simulations have been carried out till the system reached the quasi-steady state i.e., the averaged flow variables are time independent; this can be achieved by monitoring the expanded bed height or phase volume fractions.

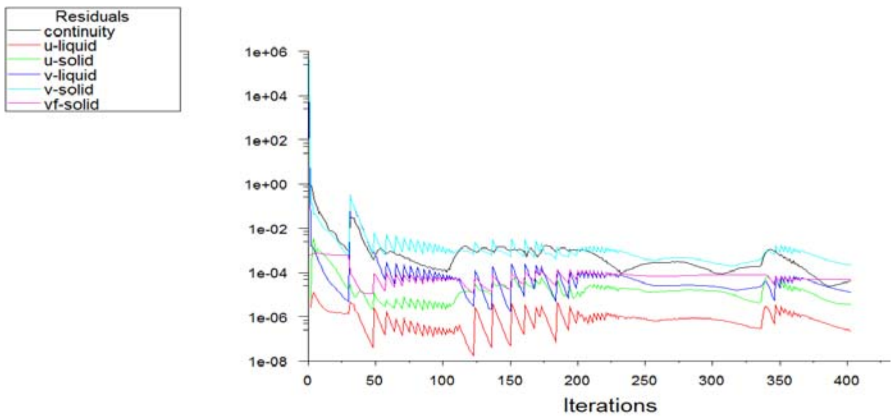
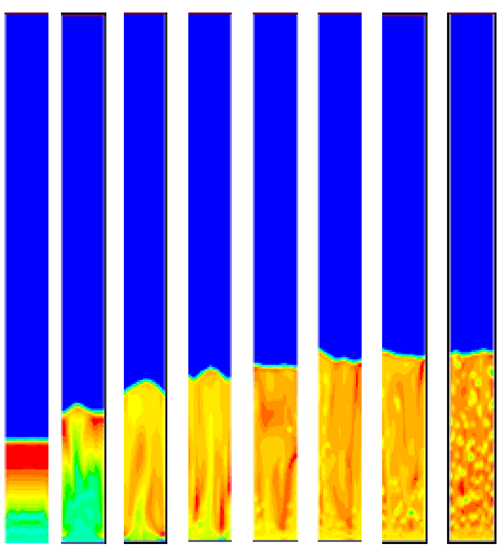


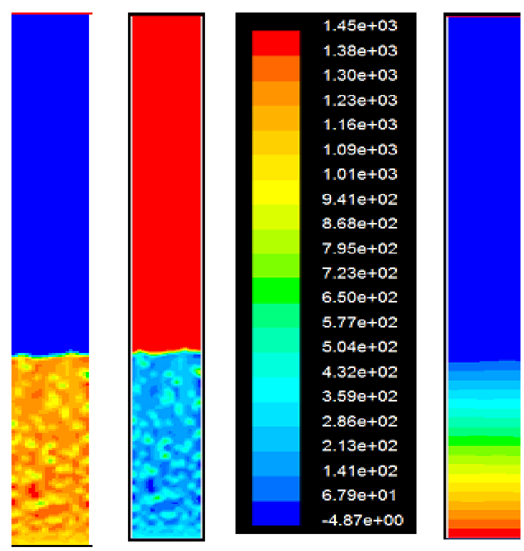
Fig. 2. Plot of residuals with the progress of simulation.

Fig. 3 shows the variation in the bed profile with time of simulation. It can be observed from the figure that, the bed profile is almost the same between 50-70 s of simulation time. Simulations were continued for 70 s and the averages over the last 20 s were used in the analysis.Once the fully developed quasi-steady state is reached, the averaged quantities in terms of time, axial and radial direction have been calculated.



5 s 10 s 20 s 30 s 40 s 50 s 60 s 70 s

Fig. 3. Contours of volume fraction of 2.18 mm glass beads at water velocity of 0.0934 m/s with respect of time for initial bed height 0.171 cm.



(a) (b) (c)

Fig. 4. (a) Contour plot of solid phase, (b) Contour plot of liquid phase, (c) Static pressure drop.

4. Results and discussion

Solid and liquid phase dynamics has been represented in the form of contours, vectors and XY plots. Fig. 4 shows the contours of volume fractions of solid and liquid in the column and the static pressure at the bottom of the column obtained at water velocity of 0.093 m/s for static bed height 0.171 m and glass beads of diameter 2.18mm after the quasi steady state is achieved. The colour scale given to the left of each contours gives the value of volume fraction corresponding to the colour. The contours for glass beads illustrates that bed is in fluidized condition.

Fig. 5 shows the velocity vectors of glass beads and water in the column obtained at inlet water velocity of 0.0934 m/s for static bed height 0.171 m and glass beads of size 2.18 m after the quasi steady state has been achieved. The velocity vectors are helpful in determining flow patterns in fluidized bed. In the upper part of the fluidizing section there is a circulatory motion of the particles, near the wall direction of velocity is downward while that in the central zone the direction is upward. The velocity vector of liquid phase in the column is mostly upward, but in the fluidized section the magnitude is more and little back mixing of the liquid is also seen in this section. The higher velocity of liquid in the fluidization section is because of less space is available for the liquid to flow. A transition from higher to lower velocity can be clearly seen at the interface between the two-phase region and pure liquid phase above it.

4.1. Bed expansion (Bed voidage)

In fluidized bed with increase in liquid velocity the bed height remains same upto the minimum fluidization condition and above the minimum fluidization velocity, the bed height increases and the voidage of the bed also increases. This phenomenon has been observed from experiments. CFD simulation result also shows an increase in bed expansion with liquid velocity. It can be seen from the contours of solid volume fraction (Fig. 6) that there is steady increase in the bed height with liquid velocity above the minimum fluidization condition. The bed height can be determined from the XY plot of the solid volume fraction w.r.t. the axial distance from the base of the column (in 2D mesh it is noted as y-coordinate) as shown in Fig. 7. The point where the solid fraction sharply decreases to zero value can be taken as the height of the bed.

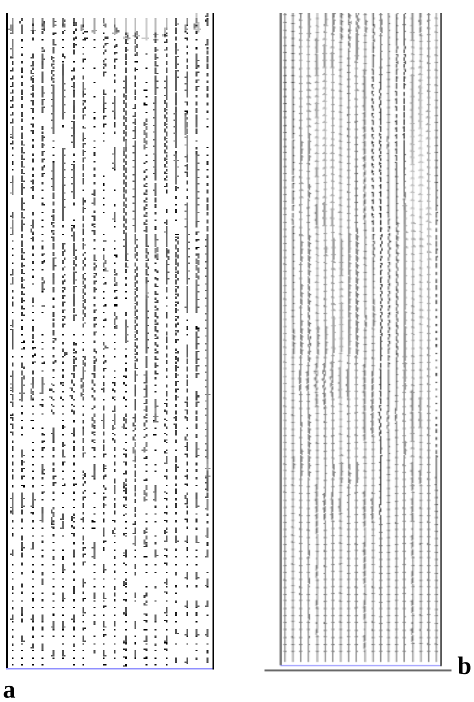


Fig. 5. Velocity vector of the (a) solid phase, (b) liquid phase.

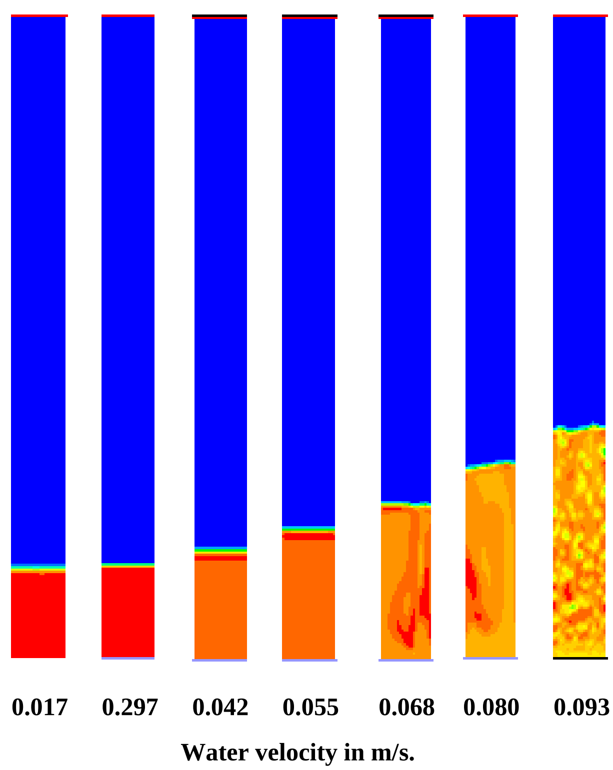


Fig. 6. Contour plot of solid volume fraction with variation in liquid velocity.

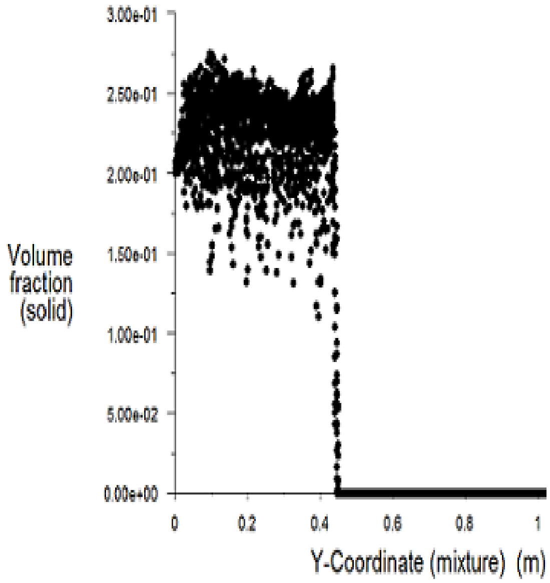


Fig. 7. XY plot of solid volume fraction.

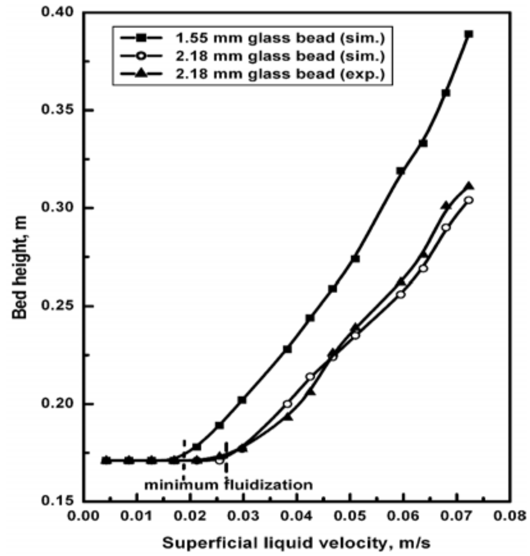


Fig. 8. Variation in height of fluidized bed with liquid velocity.

Fig. 8 shows the plot of expanded bed height vs. liquid velocity. It has been observed from the figure that the expanded bed height increases with liquid velocity. A comparison of the expanded bed height obtained from simulation has been compared with the experimental one in Fig. 8. A fairly good agreement has been observed. The fluidized bed void fraction obtained from the simulation has been compared with the calculated ones from the equation of Richardson and Zaki [12] as given below.

$$\varepsilon = \left(\frac{U_L}{U_t}\right)^{1/n} \tag{2}$$

Where the exponent 'n' is calculated as: $n = (4.4+18(d_p/D_c))\,Re^{-0.1}$ for $1<Re<200$

$n = 4.\ Re^{-0.1}$ for $200<Re<500$

$n = 2.4$ for $Re>500$

Fig. 9 presents the bed voidage values for 2.18 mm glass beads with varying liquid velocity. The figure indicates an increase in bed voidage with liquid velocity. The simulated bed voidage values agree within 20 % with those calculated from equation (2). It can be observed from Fig. 9 that the deviation increases with bed voidage value (i.e. at higher fluid velocity). It may be due to the use of a constant value of Richardson and Zaki exponent over a large range of Reynolds number.

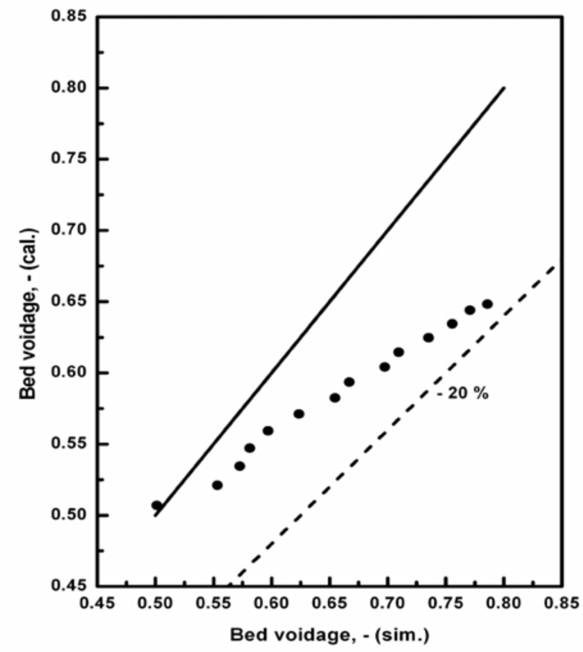


Fig. 9. Comparison of bed voidage.

5. Conlcusion

CFD simulation of hydrodynamics of liquid–solid fluidized bed has been carried out for two different particle sizes with varying liquid velocity by employing the Eulerian-Eulerian granular multi-phase approach. The CFD simulation result shows good agreement with experimental data for the hydrodynamics in term of expanded bed height (bed voidage). The bed expansion value indicates that the drag model used in CFD simulation satisfactorily describes the two-phase (liquid-solid) phenomena. The CFD simulation exhibits a solid circulation pattern for all the operating conditions, which is consistent with the observations reported by various authors. It can be seen clearly from the comparison of experimental and simulated results that Eulerian-Eulerian multi-phase granular flow approach is capable of predicting the overall bed expansion behaviour of liquid–solid fluidized bed.

Nomenclature

d_p	particles diameter, m
D_c	column diameter, m
H	bed height. m
ΔP	pressure drop, Pa
U, U_L	superficial fluid velocity, m/s
U_t	terminal velocity, m/s

Greek symbols

ε	bed porosity, -
μ	viscosity of fluid, kg/m s^2
φ	sphericity, -

References

[1] L. Mazzeia and P. Lettieri, CFD simulations of expanding/contracting homogeneous fluidized beds and their transition to bubbling, Chem. Eng. Sci. 63 (2008) 5831-5847.

[2] W. L. McCabe, J. C. Smith and P. Harriot, Unit operations of chemical Engineering, 5th Ed., (1993), McGraw Hill Int. Eds. New York.

[3] A.K. Mukherjee, B.K. Mishra, R. Vijay Kumar, Application of liquid/solid fluidization technique in beneficiation of fines, Int. J. Miner. Process. 92 (2009) 67–73.

[4] N. Epstein, Application of liquid–solid fluidisation, Int. J. Chem. Reactor Eng. 1 (2004) R1.

[5] R. Panneerselvam, S. Savithri, G.D. Surender,CFD based investigations on hydrodynamics and energy dissipation due to solid motion in liquid fluidised bed, Chem. Eng. J. 132 (2007) 159–171.

[6] A. Kumar, CFD Modeling of Gas-Liquid-Solid Fluidized Bed, B. Tech Thesis (2009) NIT Rourkela. India.

[7] S. Roy, M.P. Dudukovic, Flowmapping and modeling of liquid–solid risers, Ind. Eng. Chem. Res. 40 (2001) 5440–5454

[8] Y. Cheng, J.X. Zhu, CFD modeling and simulation of hydrodynamics in liquid–solid circulating fluidised bed, Can. J. Chem. Eng. 83 (2005) 177–185.

[9] E. Doroodchi, K.P. Galvina, D.F. Fletcherb, The influence of inclined plates on expansion behaviour of solid suspensions in a liquid fluidised bed—a computational fluid dynamics study, Powder Technol. 160 (2005) 20–26.

[10] R. Panneerselvam, S. Savithri, G.D. Surender,CFD based investigations on hydrodynamics and energy dissipation due to solid motion in liquid fluidised bed Chemical Engineering Journal 132 (2007) 159–171

[11] D. Gidaspow, Multiphase Flow and Fluidisation, 1st ed., Academic Press, San Diego, 1994.

[12] J. Richardson, W. Zaki, Sedimentation and fluidisation. Part I, Trans. IChemE 32 (1954) 35–52.