CFD analysis of hydrodynamic studies of a bubbling fluidized bed

B J M Rao[1*], K V N S Rao[2] and G Ranga Janardhana[3]

[1]Dept. of Mech. Engg., VFSTR University, Andhra Pradesh, India-522 312
[2] Dept. of Mech. Engg. B. V. R. I. T, Hyderabad, Telangana, India – 502313
[3]Dept. of Mech. Engg., JNTUA, Anantapur, Andhra Pradesh, India – 51500

*Corresponding author E-mail: bjaganmohanr@gmail.com
.

Abstract. Fluidization velocity is one of the most important parameter to characterize the hydrodynamic studies of fluidized bed asit determines different flow regimes. Computational Fluid Dynamics simulations are carriedfor a cylindrical bubbling fluidized bed with a static bed height 1m with 0.150m diameter of gasification chamber. The parameter investigated is fluidization velocity in range of 0.05m/s to 0.7m/s. Sand with density 2600kg/m^3 and with a constant particle diameter of sand 385μm isemployed for all the simulations. Simulations are conducted using the commercial Computational Fluid Dynamics software, ANSYS-FLUENT**.**The bubbling flow regime is appeared above the air inlet velocity of 0.2m/s. Bubbling character is increased with increase in inlet air velocities indicated by asymmetrical fluctuations of volume fractions in radial directions at different bed heights

Notation

ε_s	Volume fraction of particle/solid	d_s	Diameter of particle
ε_g	Volume fraction of air	ε_s^{max}	Maximum volume fraction of particle
ρ_s	Density of particle	μ_m^{max}	Maximum viscosity of particle
ρ_g	Density of air	C_D	Standard drag coefficient
$\vec{V}_s$	Velocity of solid (vector)	Re_s	Particle Reynolds number
$\overset{=}{t}_s$	Stress tensor for solid	U_{mf}	minimum fluidization velocity
$\overset{=}{t}_g$	Stress tensor for gas	u_s^{adv}	Advection velocity of solid
λ_g	Bulk viscosity of gas	u_g^{adv}	Advection velocity of gas
p_s	Solid pressure	l_{gs}	Drag force
p_f	Frictional pressure	μ_s	Solid phase viscosity
p_k	Kinetic pressure	μ_f	Frictional viscosity

p_c	Collisional pressure	μ_k	Kinetic viscosity
β_{gs}	Drag coefficient	μ_c	Collisional viscosity
g_0	Radial distribution function		
e	Coefficient of restitution		

1. Introduction

Bubbling fluidized bed is broadly utilized in industrial applicationbecause ofgood blending, heat and mass exchange. Biomass gasification reactor is one for combined heat and power (CHP) production. Flow behavior and fluidization properties in the gasifier are studied by considering the operating parameters such as pressure drop, minimum fluidization velocity and bubble behavior. These parameters significantly affect the gasifier efficiency. Understanding the hydrodynamics of fluidized bedchamber is vital for choosing the correct operating parameters for the appropriate fluidization regime [1-4] Computational fluid dynamics (CFD) can be allows to study the complicated phenomena of hydrodynamic behavior of gas and solid particles. Two models the Eulerian-Lagrangian and Eulerian-Eulerianare available in CFD for modeling of multiphase systems. Kinetic theory of granular flow (KTGF) is continuum based totally, this model is greater appropriate for simulating big and complicated commercial fluidized mattress. Fluidized bed chambers containing billions of strong particles. In precept, discrete particle models (DPM) can give such facts [5-7]. For relatively small gas feed rates, the chamber may contain a dense bed of fluidized solid particles. The bed may be homogeneously fluidized or gas may passthrough the bed in the shape of large bubbles. Further increasing the gas flow rate decreases the bed density and the gas-solid containing sample may change from dense mattress to turbulent bed, then to fast-fluidized mode and ultimately to pneumatic conveying mode. In all this flow regimes the relative importance of gas- particle, particle- particle, and wall interaction is different. It is therefore necessary to identify these regimes to select an appropriate mathematical model [5-8]. The fundamental problem in modeling of FBG is the motion of two phases which is transient, and a large number of independent variables such as particle density, size and shape can influence the hydrodynamic behavior [2, 3, 5]. Taghipour et al. [7] have conveyed exploratory and computational examinations ofsolid -gas fluidized bed chamber hydrodynamics.Their model anticipated bedexpansion and gas-solid steam designs weresensibly well and concurring with their experimental results. Huilin et al. [9] analyzed bubbling fluidized bed with parallel blends applying multifluid Eularian CFD model as per the motor hypothesis of granular stream. They found that the hydrodynamics of gas bubbling fluidized bed are related with thedistribution of particle sizes and the amount of dissipated energy in particle–particleinteractions. Gobin et al. [10] conducted numerical simulation fora fluidized bed with two-stage stream strategy. They performed time-dependent simulations for industrial and pilot chamber operating conditions. Their predictions aregood qualitative agreement with the observed behavior in terms of bed height, pressure dropand mean flow regimes. Van Wachem et al. [11] verified experimentally Eulerian-Eulerian gas-solid model simulations of bubbling fluidized beds with existing correlations for bubble size or bubble velocity using CFX, commercial CFD code. This CFD model is based on a two fluid model including thekinetic theory of granular flow. Zhonget. al [12] investigated the maximum spout able bed heights of a spout-fluid bed filled with six kinds of Geldart group D particles. The parameters investigated by them areparticle size, spout nozzlesize and fluidizing gas. They found that the maximum spoutable bed height of spout-liquid bed diminishes with increasing particle size and spout nozzle size. Lettieri et al. [13] simulated the changeover of bubbling to slugging regime for a typical Group B material at four fluidizing velocities. The simulations are carried with the Eulerian–Eulerian granular kinetic version available inside the CFX-4 code.They presented the results in terms of voidage profiles and bubble size and estimated transition velocity. Bahareh Estejab et al. [15] conducted the numerical simulations to predict the hydrodynamic behavior of gas solid mixture. Theytesteddifferent drag models. They

found the Huilin–Gidaspow drag model is suitable for single solid phase. EmbarekBelhadj et al. [16] carried out numerical simulations and experimental validation of hydrodynamics in bubbling fluidized bed combustion. The simulation methodology adopted by them is Eulerian–Eulerian approach with KTGF theory for solid particles. They validated the results with experimental data and fined to be matched.

In the presentexploration paper, hydrodynamics oftwo-measurement non-responsive solid–gas fluidizing bedchamber are concentrated computationally. A multi liquid Eulerian model consolidated the kinetic theory of solid is applied in order to simulate the gas–solid stream at various fluidized speeds.

2. Computational Methodology

ANSYS FLUENT is utilized for simulation wherein 2nd segregated first order implicit unsteady solver is utilized for multiphase calculations. The usage of Eulerian - Eulerian multiphase version, fashionable k-ε dispersed Eulerian multiphase model with standard wall functions are used. Gas is taken as continuous phase while sand particles are taken as dispersed phase. Kinetic theory of granular flow has been applied to sand particles. Gidaspow model of interphase interaction (Solid-Gas) is used. The bed material considered for the analysis is sand, an inert material. Bed fluidizing medium is air.

The continuity equation for each phase is separately as shown:

$$\frac{\partial}{\partial t}\left(\varepsilon_g \rho_g\right)+\nabla.\left(\varepsilon_g \rho_g \vec{V}_g\right)=0 \tag{1}$$

$$\frac{\partial}{\partial t}\left(\varepsilon_s \rho_s\right)+\nabla.\left(\varepsilon_s \rho_s \vec{V}_s\right)=0 \tag{2}$$

The only constraint being that total volume fraction has to add up to one.

$$\varepsilon_g + \varepsilon_s = 1 \tag{3}$$

In the present work, there is no mass transfer between the phases and thus the terms on the right hand side of the equations 1 and 2 are zero.

The gas phase momentum equation can be expressed as:

$$\frac{\partial}{\partial t}\left(\varepsilon_g \rho_g \vec{V}_g\right)+\nabla.\left(\varepsilon_g \rho_g \vec{V}_g \vec{V}_g\right)=\nabla.\overline{\overline{\tau}}_g-\varepsilon_g \nabla P+\varepsilon_g \rho_g g+\beta_{gs}\left(\vec{V}_s-\vec{V}_g\right) \tag{4}$$

Where P is the pressure, g is the acceleration due to gravity and βsg is the drag coefficient the stress tensor $\overline{\overline{\tau}}_g$ is calculated by the following equation:

$$\overline{\overline{\tau}}_g = \varepsilon_g \mu_g\left(\nabla\vec{V}_g+(\nabla\vec{V}_g)^T\right)+\varepsilon_s\left(\lambda_g+\frac{2}{3}\mu_g\right)\nabla.\vec{V}_g\overline{\overline{I}} \tag{5}$$

Assuming no virtual mass and lift force the solid phase momentum equation can be expressed as:

$$\frac{\partial}{\partial t}\left(\varepsilon_s \rho_s \overrightarrow{V_s}\right) + \nabla . \left(\varepsilon_s \rho_s \overrightarrow{V_s} \overrightarrow{V_s}\right) = \nabla . \overrightarrow{\tau_s} - \nabla P_s + \varepsilon_s \rho_s g + \beta_{gs}\left(\overrightarrow{V_s} - \overrightarrow{V_g}\right) \tag{6}$$

Where P_s is the solid pressure obtained from the Kinetic theory of granular flow? This pressure has three components kinetic, collision and frictional.

2.1 Solver and discretization scheme

The area coupled simple technique is executed for pressure–velocity coupling. The second one-order upwind scheme is hired for discretization of momentum, turbulence kinetic strength and turbulence dissipation process and the primary-order upwind scheme is executed for discretization of quantity-fraction equations. The time step of size is 0.001s is taken for the solution to converge.

2.2 Geometry and Mesh

The geometry parameters of bubbling fluidized bed reactor are primarily based at the studies work of Chinmayee Patra [14].Fig.1 (a) shows geometry of the reactor with its dimensions. The bubbling bed reactor area has inner diameters of 0.15m and height of 1m.The unfastened board region has internal diameters of 0.3m shows hydrodynamic study and height of 0.8m.The geometry is created by the use of industrial software ICEM CFD.After geometry technology, a uniform established mesh has been created as shown in Fig. 1(b).In this study, total of 39250 cells and 40086 numbers of nodes are used for simulating Fluidized Bed Gasifier. A finer mesh is adapted in the region of static bed thegenerated mesh as shown in Fig. (c)

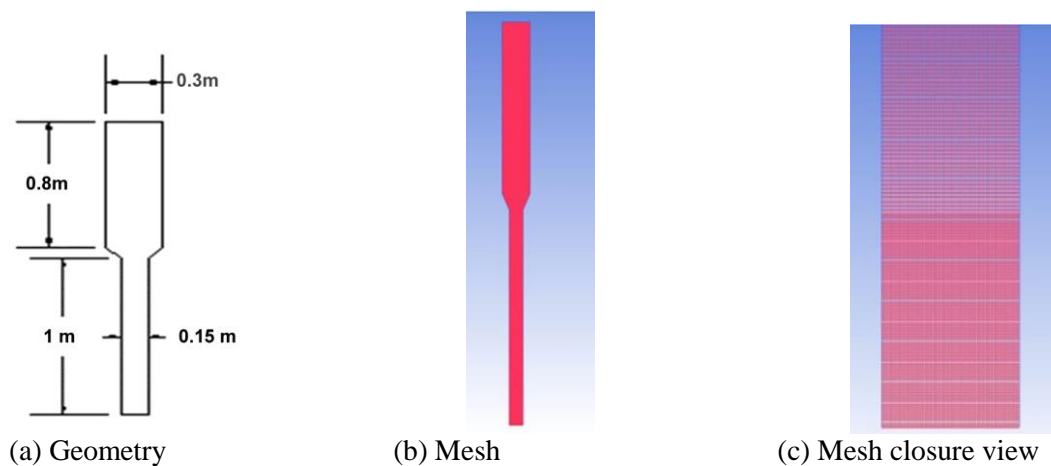(a) Geometry   (b) Mesh   (c) Mesh closure view

Fig.1, Schematic of geometry and mesh generated of bubbling fluidized Bed

3. Results and Discussion

Velocity of fluidizing media, air impact on hydrodynamic behavior of bubbling fluidized bed gasifier has been investigated using CFD simulations. The simulations are carried with CFD commercial software ANSYS FLUENT. The results in the form of contour plots, vectors and chats have been presented below for solid volume fraction, phase velocity.

Fig.2 and 3 represents the solid volume fraction contours of sand for the different air velocities i.e. at 0.05m/s, 0.2 m/s, 0.5m/s and 0.7 m/s complete and closure view at bed zone.At the velocity of 0.05m/s the bed begins to expand on a small scale but almost the height of the bed is equal to static bed height. For the air velocity of 0.2m/s, bubbles are appeared like surface only and they found in the static bed height without any appreciable bed growth. The reason attributed may be sand debris within the bottom part of the bed are in pneumatic transport while fluidization within the upper portion is in freely bubblingstate. At the air velocity of 0.5m/s and 0.7m/s, it is observed from contour the bubbles flow regime is appeared. This may be reason of the segregate tendencies of the particles towards the partitions or gulf streaming. For that reason, the solid particles slide down along the wall of the reactor without too much resistance from the upward gas flow.
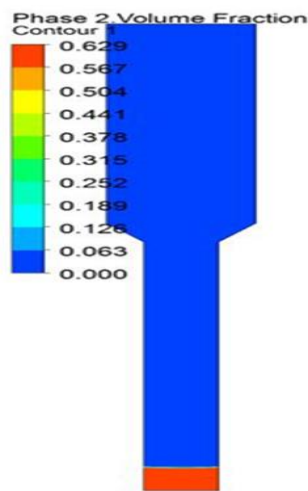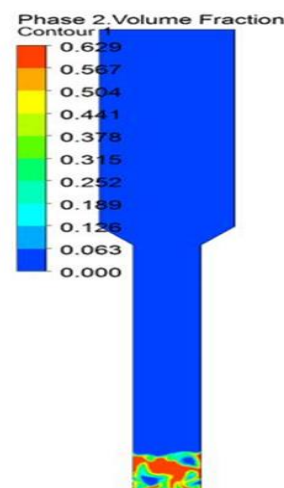

a) 0.05m/s
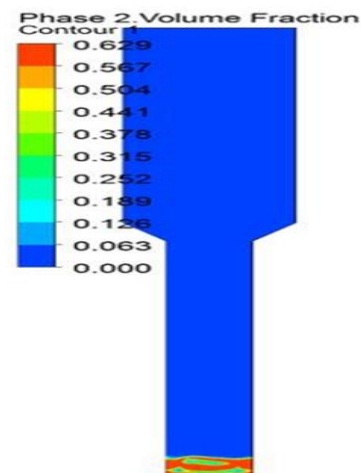
b) 0.2m/s
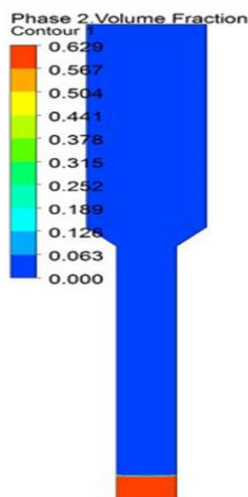
c) 0.5m/s

d) 0.7m/s

Fig. 2, Sand volume fraction contour at different air velocities for initial static bed height of 0.1m complete chamber

a) 0.05m/s
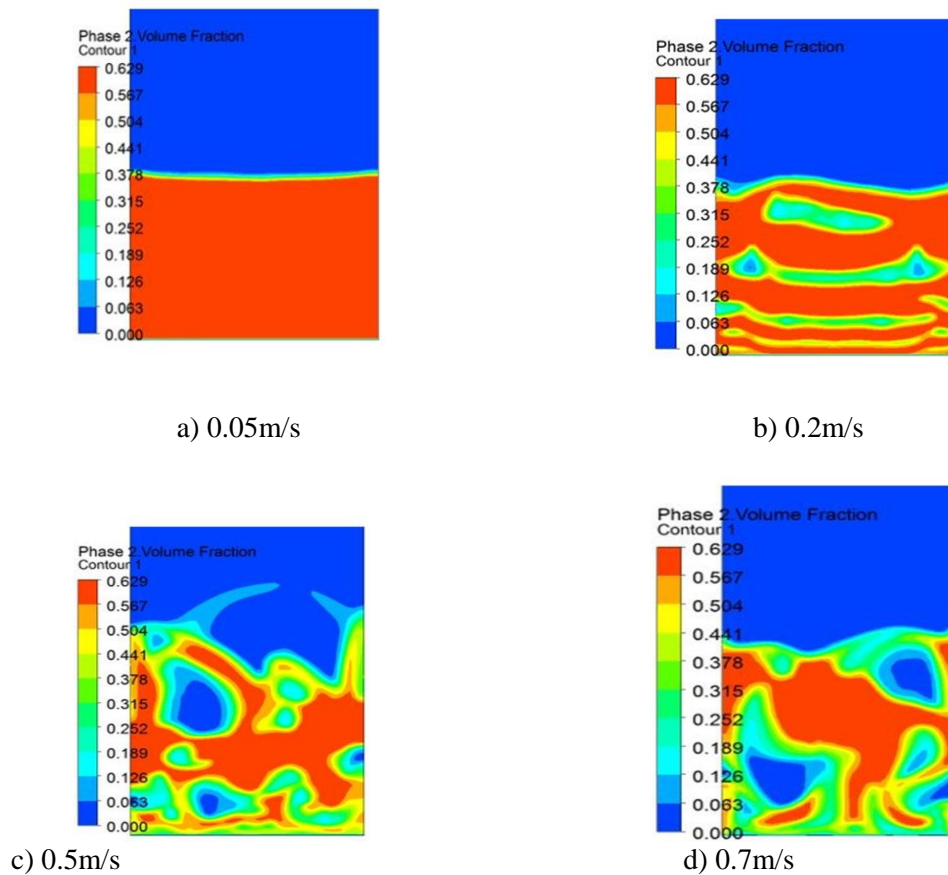

b) 0.2m/s

c) 0.5m/s

d) 0.7m/s

Fig. 3, Sand volume fraction contour at different air velocities for initial static bed height of 0.1m near the bed zone.

Flow pattern in fluidized bed is shown in form of the velocity vectors for sand in the Fig.4 for different air velocities. It is noticed that at lower velocities the solid phase velocity is more at the upper surface and lower at the bottom i.e. at 0.05m/s as the velocity increased to 0.2m/s, sand is appeared to be lifting in form of surfaces. The bubbling regime is come in to sight at the velocity of 0.5m/s. At the velocity 0.7m/s, this is noticed that there was an aggressive movement of solid particles throughout the bed implying that the velocity at the bottom is less.

a) 0.05m/s

b) 0.2m/s

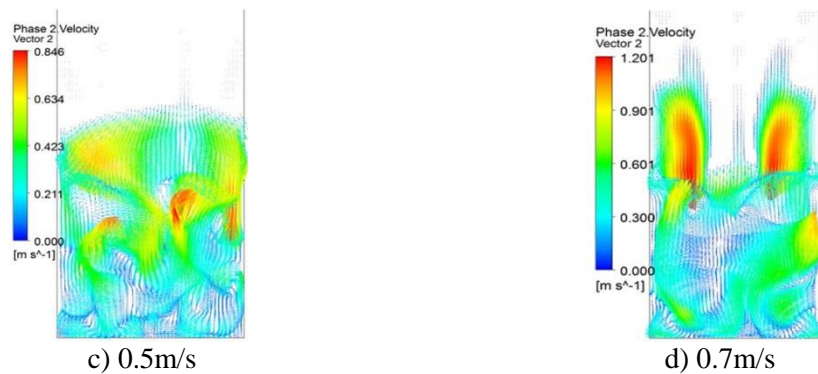c) 0.5m/s   d) 0.7m/s
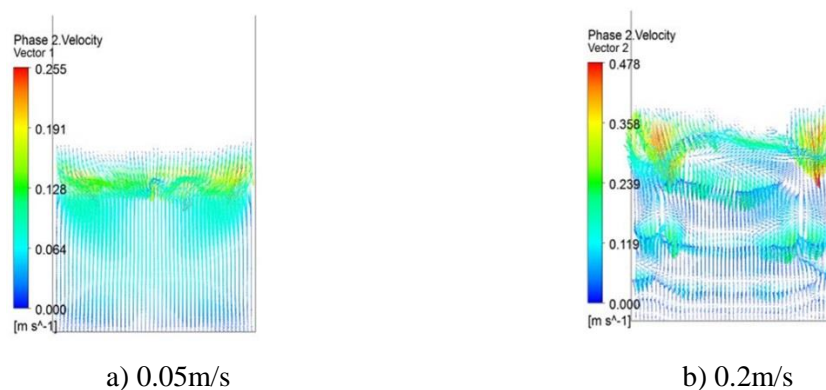
Fig.4, Velocity vector of sand at different air velocities

Fig.5 and 6 represents the complete and closure views of gas phase velocity vectors in the chamber. The gas phase's velocity vectors in the chamber are heading upwards and velocity is found to be less at the region, higher solid volume fraction as the obstruction was more. For the low velocity 0.05m/s, the air is not able to penetrate properly through the sand bed. At the velocity 0.2m/s, it becomes more and increases further with increase in air velocity.
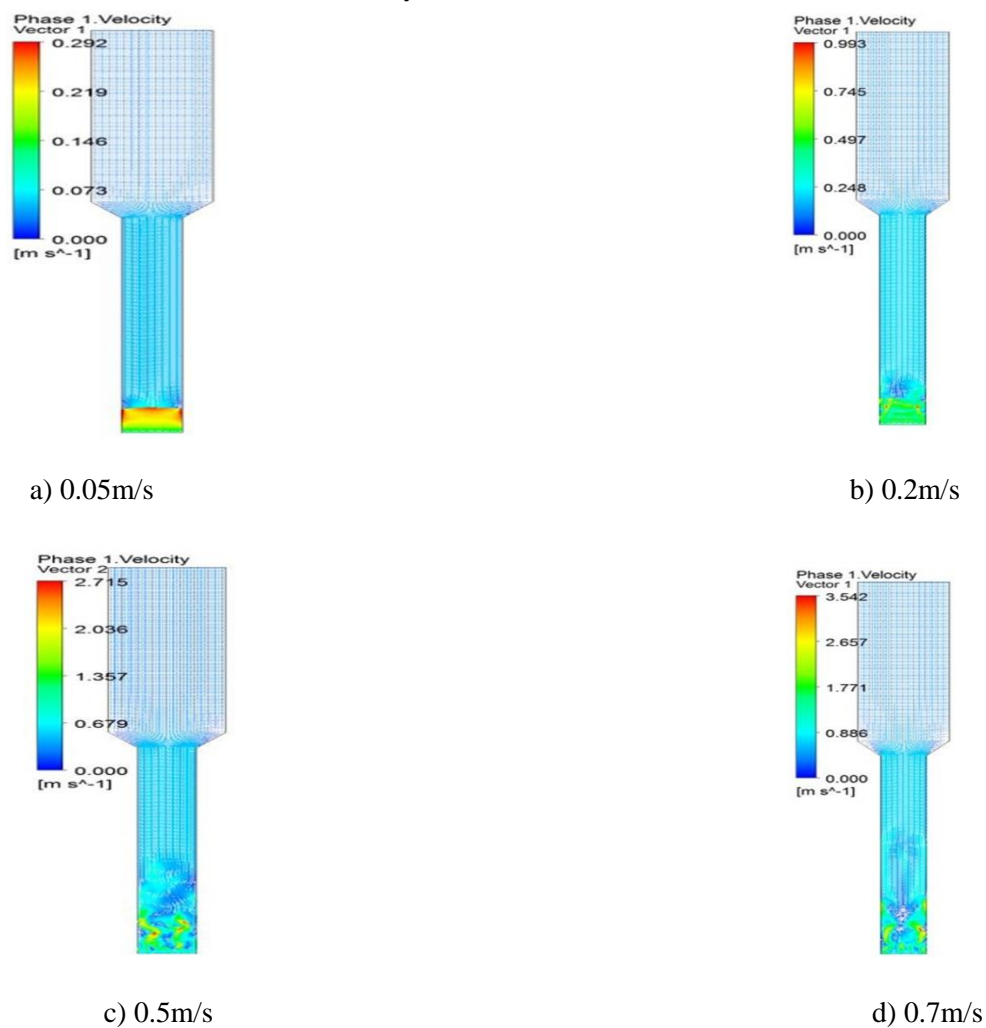

a) 0.05m/s   b) 0.2m/s

c) 0.5m/s   d) 0.7m/s

Fig.5, Velocity vector of air at different air velocities in complete chamber

a) 0.05m/s   b) 0.2m/s

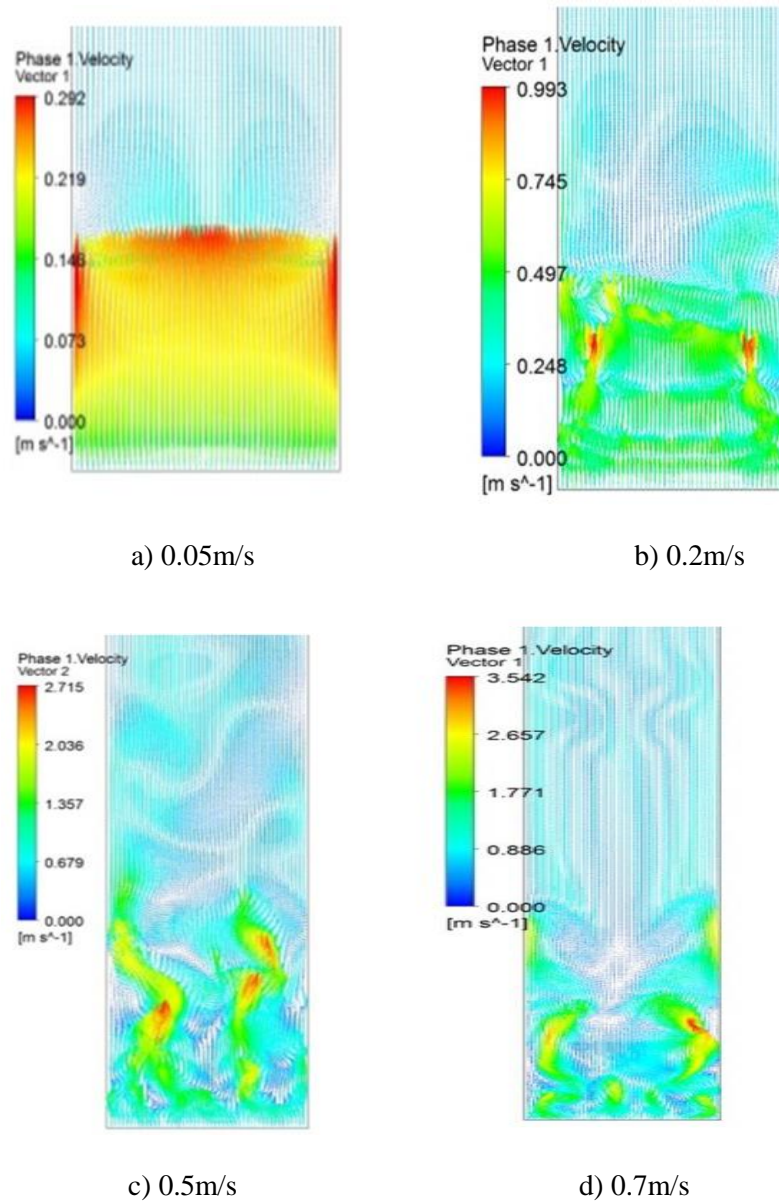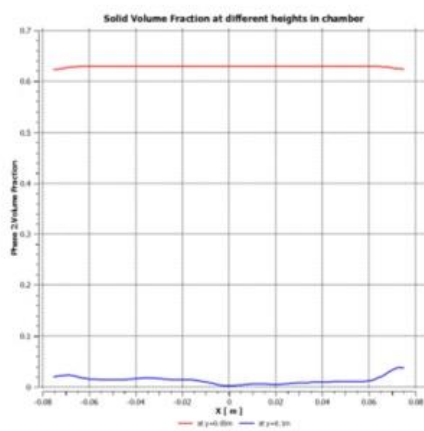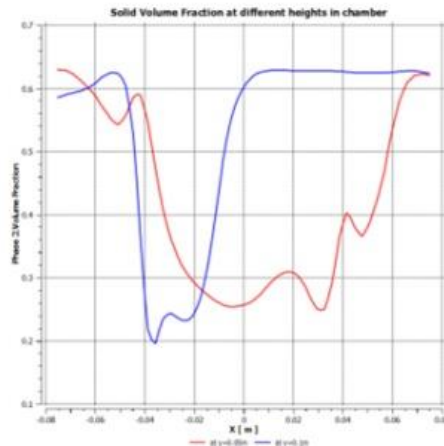c) 0.5m/s   d) 0.7m/s

Fig.6, Velocity vector of air at different air velocities in complete chamber at closure view

Phase 2 i.e. sand volume fractions are plotted against radial length at different axial heights as shown in Fig.7. At air velocity 0.05m/s the bed expansion is at small scale only two heights are considered 0.05m and 0.1m. The volume fraction at 0.05m is higher and almost constant in radial direction indicating that low air penetration. In addition small scale expansion is indicated at 0.1m bed height with slight fluctuation in sand radial distribution. As the velocity is increased to 0.2m/s sand volume fraction at the central core region is less as compared to the wall which indicating the air penetration, which shows higher particle volume fractions along the walls as compared to the core region. With further increase in velocities to 0.5m/s and 0.7m/s the bubbling regime appeared as it is indicated by asymmetrical fluctuations of volume fractions in radial directions at different bed heights. From the simulation result as shown in the Fig.2. The hydrodynamic model is able to describe quantitatively the accumulation of solids near the wall.

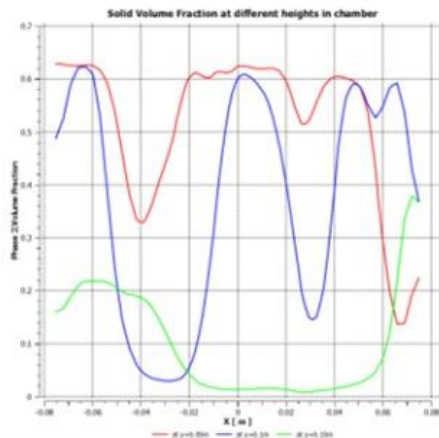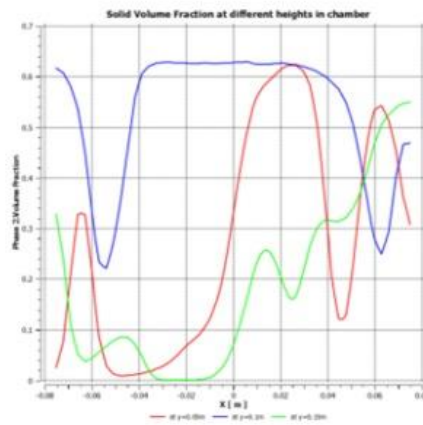a) 0.05m/s   b) 0.2m/s
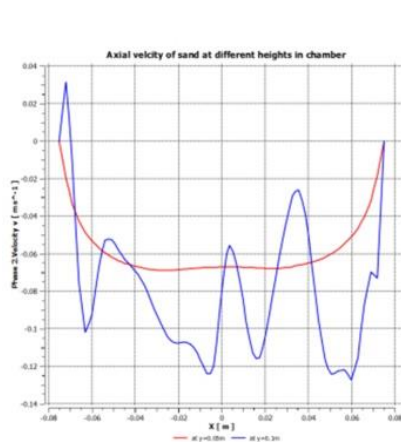

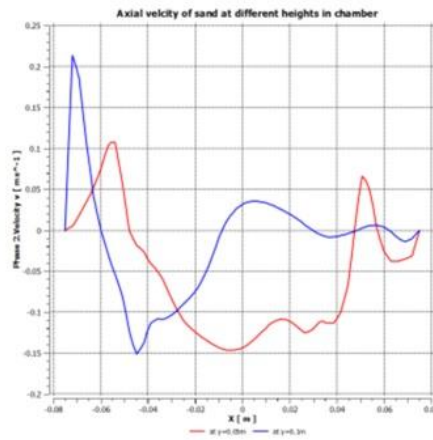c) 0.5m/s   d) 0.7m/s

Fig. 7, sand particle concentration against the radial position for different bed heights at different air inlet velocities
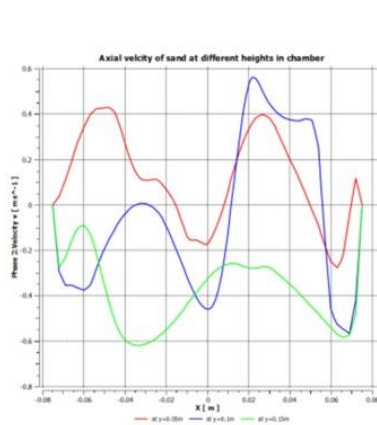
Fig.8 represents the plot of radial variation of axial velocity of sand particles in chamber at different bed heights for different air inlet velocities. Sand particle velocities near wall are decreases and this may lead to the back accumulation of particles.
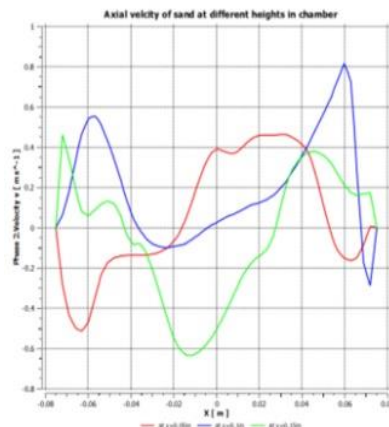
a) 0.05m/s   b) 0.2m/s

c) 0.5m/s   d) 0.7m/s

Fig. 8,Sand particle velocity distribution along the radial direction at different velocities for different axial heights of 0.05 m, 0.1 m,0.15 m and 0.2m

4. Conclusions

CFD simulations using ANSYS-Fluent software carried out to investigate hydrodynamic behavior of bubbling fluidized bed. The bubbling flow regime is appeared above the air inlet velocity of 0.2m/s. Bubbling character is increased with increase in inlet air velocities indicated by asymmetrical fluctuations of volume fractions in radial directions at different bed heights.

References

[1]   Gidaspow D, Multiphase Flow and Fluidization, First ed, London.1994.
[2]   Kunii D, Levenspiel O, Fluidization Engineering, Second ed., Butterworth- Heinemann, Boston 1991.

[3]   Ranade V V, Computational Flow Modeling for Chemical Chamber Engineering, Firsted. Academic press, New York.2002.

[4]   Huilin L, Wentie L, Feng L, Guangbo Z, Huilin H L, Wentie L, Feng L, Guangbo Z, Yurong H, Eulerian. Simulations of bubble behavior in a two dimensional gas–solid bubbling fluidized bed, *Int. J. of Energy Research*, 26, pp.1285–1293. (2002)

[5]   Grace J R, Taghipour F, Verification and validation of CFD models and dynamic similarity for fluidized beds. *Powder Technology,* pp. 139, 99–110.2004

[6]   Bird R B, Stewart W E, Lightfoot E N, *Transport Phenomena,* Wiley.2002.

[7]   Taghipour F, Ellis N, Wong C, Experimental and computational study of gas–solid fluidized bed hydrodynamics, *Chemical Engineering Science*, 60, pp. 6857–6867.(2005)

[8]   Kaneko Y, Shiojima T, Horio M, DEM simulation of fluidized beds for gas-phase olefinpolymerization, *Chemical Engineering Science*, 54, pp. 5809–5821.1999.

[9]   Huilin L, Yurong H, Gidaspow D. Hydrodynamic modeling of binary mixture in a gas bubbling fluidized bed using the kinetic theory of granular flow, *Chemical Engineering Science*, pp. 1197–1205, 2003

[10]   Gobin H, Neau O, Simonin J, Llinas V, Reiling J L, Selo, Fluid dynamic numerical simulation of a gas phase polymerization chamber, *International Journal for* NumericalMethods in Fluids, 43, pp. 1199–1220. 2003

[11]   Van Wache B G M, Schouterf J C, Krishna R, Bleek vanden C M, Eulerian Simulations of Bubbling Behaviour in Gas-Solid Fluidized Beds, *Computers chem.Eng.* , Vol.22, pp. 299-306.1998

[12]   Zhong W, Zhang M, Jin B, Maximum spoutable bed height of spout-fluid bed, *Chemical Engineering*, 124, pp. 55–62.2006

[13]   Lettieri P, Saccone G, Cammarata L, Predicting the Transition from Bubbling to Slugging Fluidization Using Computational Fluid Dynamics ,*Trans IChemE, Part A, Chemical Engineering Research and Design*, 82(A8): 939–944., 2004

[14]   Chinmayee Patra, CFD simulations of fluidized bed biomass gasification, A Thesis Submitted to the National Institute of Technology, Rourkela In Partial Fulfillment for the Requirements Of Master of Technology (Res.) Degree

[15]   Bahareh Esteja, Francine Battaglia, Assessment of Drag Models forGeldart A particles in BubblingFluidized Beds, *Journal of Fluids Engineering*, 138(2015)

[16]   Embarek Belhadj, Stephen Chilton Numerical simulation and experimental validation of the hydrodynamics in a 350 kW bubbling fluidized bed combustor, *Int. J. Energy Environ Eng* 7 (2016):27–35