NUMERICAL INVESTIGATION OF HYDRODYNAMICS AND HEAT TRANSFER IN A PRESSURIZED CIRCULATING FLUIDIZED BED RISER

Abinash Mahapatro[1*], Pinakeswar Mahanta[1]

[1]*Department of Mechanical Engineering, Indian Institute of Technology, Guwahati, 781039, India*

*Email: *abinashlipun@iitg.ernet.in*

Abstract – Present work includes numerical investigation on bed hydrodynamics and heat transfer in a pressurized circulating fluidized bed (PCFB) unit. Coal with mean particle diameter of 407 μm and density of 1350 kg/m^3 is used as the bed material. Eulerian-Eulerian phase model with Gidaspow drag model is used for phase interaction where air is considered to be the gas phase. Mass, Momentum and Energy conservation equations for both the phases are solved by finite volume method. The flow model is simulated using a well-established commercial software ANSYS FLUENT-14. All the boundary conditions are applied based on experimental data. A two-dimensional, double-precision pressure-based, absolute velocity formulation with transient model solver is used for simulation of both gas and solid phase. The second order upwind scheme is applied in the spatial discretization method except volume fraction as QUICK. The effect of input pressure on bed voidage, suspension density, and temperature profile has been studied and compared with available experimental data.

INTRODUCTION

Pressurized circulating fluidized bed (PCFB) is the third generation fluidized bed which is attracting the researchers for generation of power in future due to its compactness and high bed-to-wall heat transfer capability. It has got extensive applications in generation of power as well as in chemical industries. Bed hydrodynamics, being closely associated with the heat transfer phenomenon, plays an important role in designing the PCFB.

Shen *et al.* (1991) experimentally investigated the effects of bed voidage in a PCFB unit. Yates (1996, 1997) studied the effect of pressure and temperature on gas solid fluidization on Geldart A and B particles and found that the minimum fluidization velocity, the bubbling velocity and the terminal velocity decreases with increase in pressure. Similarly Kalita *et al.* (2013) had reported an increase in suspension density at an elevated pressure for PCFB. Both Kalita *et al.* (2013) and Nag and Gupta (1999) reported an increase in axial bed voidage at the bottom of the bed and decrease of the same at the top zone with increase in operating pressure. Esmail et.al. (2015) observed the exit configuration of the riser pressure of a CFB riser working beyond fast fluidization regimes for three different materials that are cork, coke and the glass beds and they found T shaped voidage profile at the riser exit and the voidage at the exit is linearly related to the solid flux and density of the gas and solids. Khurram *et al.* (2016) investigated the relationship between solid circulation rate and pressure drop of pressurized circulating fluidized bed riser where they found that the pressure drop increases as the solid circulation rate or the particle size increases.

Richtberg *et al.* (2005) conducted some experimental investigations in a 0.19 m diameter and 9 m high pilot scale PCFB unit in order to characterize the flow patterns. Information obtained is used to develop a correlation for the prediction of internal solids reflux in a riser reactor as a function of solids/gas density ratios and the dimensionless superficial gas velocity.

Winkya and Basu (2001) investigated the bed to wall heat transfer in a PCFB with 52.5 mm riser diameter and 2020 mm riser height. They reported that heat transfer coefficient increases with increase in operating pressure and bed temperature due to the mixed effect of gas convection and radiation. Blaszczuk *et al*. (2016) examined the heat transfer coefficient of the fluidized bed with different particle size and found that for the smaller particle convection component is dominant whereas radiation component plays significant role in case of larger particle.

Further various researchers have numerically analyzed different aspects of bed hydrodynamics and heat transfer for ACFB boiler by implementing turbulence models as described by Peng *et al.* (2011, 2011). Benyha *et al.* (2000) simulated the riser section of an ACFB where Syamlal O'Brien model is used as the drag model and k - ε model as turbulence model. They have successfully estimated the velocity, volume fraction, pressure and turbulence parameters for each phase. Neri and Gidaspow (2000) as well as Shah *et al*. (2012) have used the kinetic theory of granular flow to define the fluid properties of the solid phase through constitutive equations. They have simulated the riser section to obtain the time averaged data for particle concentrations and fluxes which confirms the existence of core annulus flow regime. Mahapatro et al. (2014) simulated the riser using Ansys fluent for the PCFB riser of 2 m in height and 0.054 m in diameter. They have compared their numerical simulation result with published data for bed hydrodynamics under the pressure range of 1 bar to 15 bar. However, literature on numerical simulation of a PCFB riser in investigating hydrodynamics is found to be limited.

In the present investigation, an attempt has been made to investigate the 2-D bed hydrodynamics in a PCFB riser by using ANSYS FLUENT 14. Riser of height 2 m and diameter 0.054 m is considered in the present investigation. Air is considered as primary fluid whereas coal with mean particle diameter of 407 μm is considered as secondary fluid. Present simulation considered as operating pressure from 1 bar to 10 bar with 500^0 C input air temperature. The mass, momentum and energy conservation equations for each phase are solved using finite volume approach. An Eulerian-Eulerian model is considered for the multi-phase flow simulation. The simulation is carried out using ANSYS FLUENT 14. Results found in the present simulation are compared with the published data of Mahapatro *et al*. (2014).

Simulation conditions

For obtaining numerical results, conservation of mass and momentum equations were used. The conservative equations were solved by using finite volume approach with the help of ANSYS FLUENT 14. Dispersed turbulence multiphase model was used with the k - ε model. The mass conservation, momentum and energy equation reported by Shah *et al.* (2012) for solid and gas phase has been used for the present simulation. The riser section of a PCFB used in numerical simulation of the multiphase flow is shown in Fig.1. The riser is 0.054 m in diameter and 2 m in height. The boundary conditions are as velocity inlet and pressure outlet as inlet and outlet respectively whereas for returning of the coal from the down comer, mass flow inlet boundary condition is considered. Simulation was performed for both the phase's viz. air and solid. The solid particles used in the simulation consist of coal with average diameter of 407 µm and density of 1350 kg/m^3. Simulations were performed with constant superficial velocity of 6 m/s by varying the operating pressures from 1 bar to 10 bar.

The geometry of riser and the properties of solid and gas considered in the present investigation are presented in Table 1.

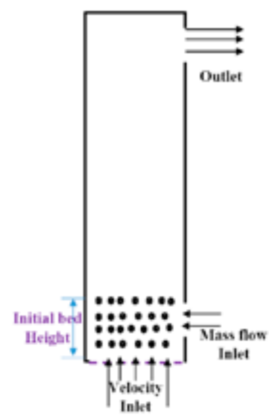

Fig. 1. Schematic drawing of the simulation model

Table 1: Basic Geometry and properties

Geometry and properties	Value
Riser diameter , *D (m)*	0.054
Riser height, *H (m)*	2
Particle size, $d_p\left(\mu m\right)$	407
Gas density, $\rho_g\left(kg\,m^{-3}\right)$	1.225
Solid density, $\rho_s\left(kg\,m^{-3}\right)$	1350
Gas viscosity, $\mu_g\left(pas\right)$	1.000000

Numerical solution methods

For simulation work Ansys design modeler and mesh editor has been used for creation of geometry and meshing respectively. All the computational variables are presented in Table 2. The 2-D computational domain consists of 14 grids radially and 230 grids axially. A total number of 3228 cells are used for grid distribution in this simulation work. All the governing equations are solved by the finite volume approach. Phase coupled SIMPLE scheme is used in the pressure-velocity coupling for gas solid flow in the algorithm. Second order upwind scheme is applied in the momentum discretization method except volume fraction is applied as QUICK scheme. A time step of 0.001s is used with 20 iterations per time step. The simulation is continued till 360 min real simulation time to reach the steady state condition.

Table 2: Simulation parameters

Simulation parameters	Value
Viscous model	Turbulent
Under-relaxation parameter	
Pressure	0.5
Momentum	0.4
Granular temperature &Volume fraction	0.2
Discretization	
Momentum and Granular temperature	Second order upwind
Maximum particle volume fraction	0.63

Boundary conditions

Inlet boundary condition was considered to be constant superficial velocity with air temperature of 500^0 C. For pressurized the bed operating pressure of the bed is increased from 1 bar to 10 bar for different simulation conditions. The outlet boundary condition is as pressure outlet with atmospheric pressure at the outlet. For the return leg, where coal particles returned to the riser are considered as mass flux inlet with constant value by solving the mass conservation equation. In wall boundary condition Johnson-Johnson is applied conditions consisting of no-slip condition for the gas phase and specified shear condition for the solid phase were considered. The quantative data for all the boundary conditions are given in Table 3.

Table 3: Boundary conditions

Type	Conditions	Value
Inlet	Gas superficial velocity *(m/s)* Solid volume fraction Granular temperature Mass flux *(kg/m^2s)*	6 0.54 0.01 35.65
Outlet	Pressure outlet Gauge pressure *(Pa)* Backflow granular temperature (m^2s^{-2}) Backflow volume fraction	0 0.01 0.01
Wall	Specularity coefficient φ Particle-particle restitution coefficient e_s	0.0005 0.95

Results and Discussion

In the present investigation, the bed hydrodynamics and heat transfer of a PCFB unit is investigated using a two dimensional geometry of height 2 m in ANSYS FLUENT 14. The variations of Suspension density is compared with Mahapatro *et al.* (2014).

Grid independency Test

In order to confirm the CFD results to be independent of mesh size, simulations were performed at three different mesh sizes viz. 1614 cells, 3228 cells and 6456 cells. The computed pressure drop along the height of the riser for different cell sizes are plotted in Fig. 2. From the figure, it can be concluded that for the cells sizes of 3228 and 6456, the correct pressure drop could be predicted. However, the computational time required for 6456 cells sizes is 1.4 times more than that of 3228 cells. The cells size of 3228 is found suitable for providing a reasonably mesh independent result at expense of less computational time. Hence, this mesh size is considered in the present investigation.

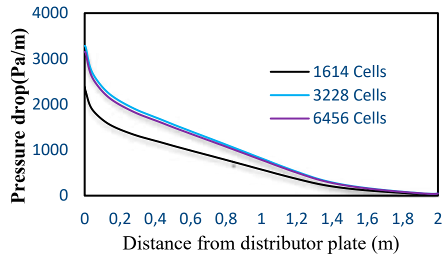

Fig. 2 Grid independency test

Effect of pressure on bed voidage

The comparison of bed voidage along the height of the PCFB riser at operating pressures of 1 to 10 bar respectively is shown in Fig. 3. The bed voidage profile is obtained using Gidaspow drag model with a velocity of air as 6 m/s, coal inventory of 600 g and with particle size of 407µm.

It is observed from the figure that the voidage increases along the height of the bed. However, voidage is observed to be less in the bottom of the bed. It is also observed that bed voidage decreases with increase in operating pressure. A phase shift is prominent for minimum voidage with increase in bed pressure. This indicates the dominance of pressure drag over the frictional drag. Bed voidage profile is flatter at high pressure (8 and 10 bar) same exhibits the typical S profile at relatively low operating pressure (1 to 5 bar).

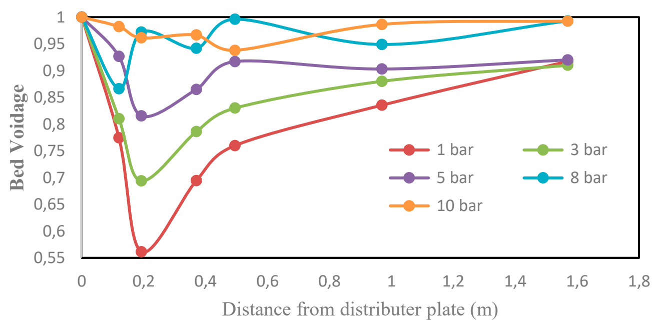

Fig. 3 Variation of bed voidage with different pressure for present simulation

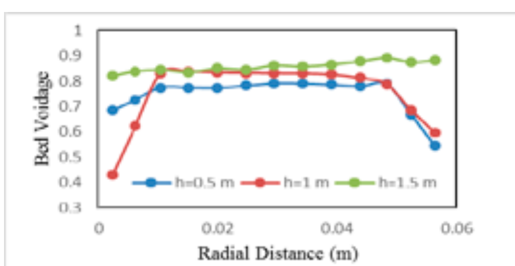

Fig. 4 Radial voidage distribution at different height of the riser

Figure 4 presents variation of bed voidage with radial distance at different locations of the bed. This plots are made for axial position 0.5 m, 1m and 1.5 m measured from the distributor plate. Voidage profile is observed to be flat in the core. However voidage decreases near to the wiser wall. This is because of the increase in solid concentration near the wall.

Effect of pressure on suspension density

Fig. 5 shows suspension density with different system operating pressure from 1 bar to 10 bar along the height of the riser. The suspension density of the bed (ρ_{sus}) is defined by (Kunii and Levenspiel, 1991) as $\rho_{sus} = \rho_s(1-\varepsilon)+\varepsilon\,\rho_g$. The suspension density is made as custom field function in ANSYS FLUENT 14, and after the convergence of code the profile is obtained for different pressure. It is observed that at higher pressure the suspension density is more at the middle and top part of the riser .It is noticed that suspension density increases as pressure increases along the height of the PCFB riser.

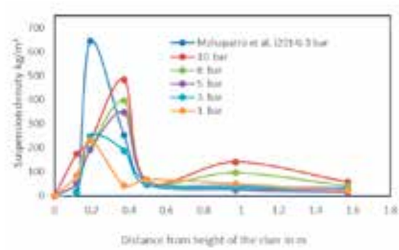

Fig. 5 Variation of suspension density at different pressure for present simulation

The present simulation data of suspension density for operating pressure of 3 bar with coal particles as inventory is compared with Mahapatro et al. (2014) where sand was used as inventory. The suspension density is comparable with Mahapatro et al.(2014) bur the difference in value is due to the type of inventory used in both the simulations.

Effect of pressure on temperature profile

The temperature variations along the bed height at a superficial velocity of 6 m/s and at an operating pressures of 1 bar to 10 bar for 600 g bed inventory with input air temperature at 500^0 C is shown in Fig. 6. It has been found that the bed is not isothermal. The maximum temperature is recorded at 0.75 m above the distributor. Then it tapers off along the height of the riser. It has been observed the bed temperature increases with higher inlet pressure. Due to the proper mixing of solid particles at higher pressure a uniform variation of temperature has been observed.

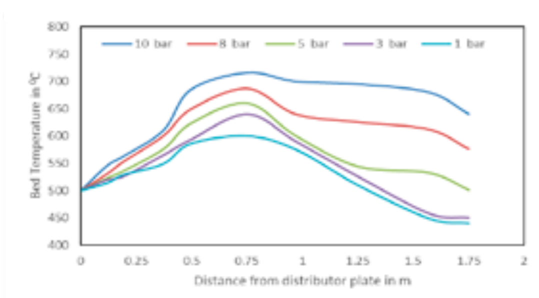

Fig. 6 Variation of Bed temperature at different pressure for present simulation

CONCLUSIONS

From the present simulation, it can be concluded that the numerical results for suspension density along the height of the riser at an operating pressure of 3 bar are comparable with the earlier published results. Flatten S-Shaped bed voidage profile along the height of the riser has been observed. An increase of suspension density has been found with increase in operating pressure. The bed temperature increases with increase in operating pressure. The results obtained in the study will be helpful for scale-up and design of a PCFB riser.

Acknowledgement

The financial support provided by Central Power Research Institute, Bangalore for sponsoring the project on pressurized circulating fluidized bed unit is gratefully acknowledged.

References

Almuttahar, A., Taghipour, F., 2008, "Computational fluid dynamics of high density circulating fluidized bed riser: Study of modeling parameters". Powder Technology, 185(1), pp.11-23.

Blaszczuk, A., Zylka, A. and Leszczynski, J., 2016. Simulation of mass balance behavior in a large-scale circulating fluidized bed reactor. *Particuology*, *25*, pp.51-58.

Benyahia, S., Arastoopour, H., Knowlton, T.M., Massah, H., 2000."Simulation of particles and gas flow behavior in the riser section of a circulating fluidized bed using the kinetic theory approach for the particulate phase" Powder Technology, 112, pp.24–33.

Cheng Leming and Basu P. 1999, "Effect of pressure on loop seal operation for a pressurized circulating fluidized bed", Powder Technology, 103, pp.203-211.

Gupta, A.V.S.S.K.S., Nag, P.K., 2002, Bed-to-wall heat transfer behavior in a pressurized circulating fluidized bed, Int. J. Heat Mass Transfer, 45, pp. 3429–3436.

Kalita, P., Saha, U.K., Mahanta, P., 2013. "Parametric study on the hydrodynamics and heat transfer along the riser of a pressurized circulating fluidized bed unit", Experimental Thermal and Fluid Science, 44, pp. 620–630.

Kallio, S., Gulden, M., Hermanson, A., 2009."Experimental study and CFD simulation of a 2D Circulating Fluidized Bed".Proceedings of the 20th International conference on Fluidized Bed combustion, May 18-21, pp.799-804.

Khurram S, Choi J H, Won Y S , Ryu H J. 2016. Relation between solid flow rate and pressure drop in the riser of pressurized circulating fluidized bed. Journal of chemical engineering of japan, vol 49, No.7, pp. 595-601 2016.

Kunnii, D., Levenspiel, O., 1991.Fluidization engineering, second ed. Butterworth-Heinemann, USA.

Li, J.J., Zhang, H., Yang, H.R., Wu, Y.X., Lu, Yue, J.F., Zhang, G.X., 2009, "Hydrodynamic model with binary particle diameter to predict axial voidage profile in a CFB combustor", in: Proceedings of the 20th International Conference on Fluidized Bed Combustion, pp. 768–773.

Mahapatro A, Kalita P, Mahanta M, Saha U. K. , Mallick S. S., 2014. Numerical simulation of gas-solid flow in a pressurized circulating fluidized bed riser, Proceedings of the 11th International conference on Fluidized Bed Technology (CFB-11), pp. 323-328, May 14-17, 2014, Beijing, China.

Molerus, O., 1993. "Arguments on heat transfer in gas fluidized beds", Chemical Engineering Science, 48 (4), pp.761-770.

Nag, P.K., Gupta, A.V.S.K.S., 1999, "A heat transfer model of pressurized circulating Fluidized bed in: J.werther(Ed.)". Circulating Fluidized bed Technology, 6, pp.361-366.

Neri, A., Gidaspow, D., 2000. "Riser hydrodynamics: simulation using kinetic theory".AIChE Journal, 46, pp.52–67.

Peng, B., Xu, J., Zhu, J., Zhang, C., 2011, "Numerical and experimental studies on the flow multiplicity phenomenon for gas–solids two-phase flows in CFB risers".Powder Technology, 214, pp.177–187.

Peng, B., Zhang, C., Zhu, J., 2011, "Numerical study of the effect of the gas and solids distributors on the uniformity of the radial solids concentration distribution in CFB risers". Powder Technology, 212, pp. 89–102.

Plasynski S. ,Klinzing G., Mathur M., 1994, "High pressure vertical Pneumatic transport investigation", Powder Technology., 798, pp.95-109.

Reddy, S.B.K., Knowlton, T.M., 1996,"The effect of pressure on CFB riser hydrodynamics", Proceedings of 5th International Conference on CFB, Beijing.

Shah, S., Ritvanen, J., Hyppänen, T., Kallio, S., 2012. "Space averaging on a gas–solid drag model for numerical simulations of a CFB riser",Powder Technology, 218, pp.131–139.
Shen, X., Zhou, N., Xu, Y., 1991, Experimental study on heat transfer in a pressurized circulating fluidized bed, in: P. Basu, M .Hasatani (Eds.), Circulating Fluidized Bed Technology III, Pergamon Press, New work, pp. 451-456.
Tang, J.T., Engstrom, F., 1987, "Technical assessment on the Ahlstrom pyro flow circulating and conventional bubbling fluidized bed combustion systems", in: J.P. Mustonen (Ed.), Proceedings of the 9th International Conference on Fluidized Bed Combustion, ASME, New York, pp. 38–54.
Winaya N.S., and Basu P., Effect of pressure and carbon dioxide concentration on heat transfer at high temperature in a pressurized circulating fluidized bed (PCFB) combustor, International Journal of Heat and Mass Transfer. 44 (2001) 2965-2971.
Yates, J.G., 1996, "Effects of temperature and pressure on gas-solid fluidization", Chemical Engineering Science, 51(2), pp.167 -205.
Yates, J.G., 1997, Experimental observations of voidage in gas fluidized beds, in: J. Chaouki, F. Larachi, M.P. Duducovic (Eds.), Non-Invasive Monitoring of Multiphase Flows, Elsevier, Amsterdam, pp. 141–160.