International Journal of Chemical Engineering and Applications, Vol. 1, No. 2, August 2010
ISSN: 2010-0221

Experimental Studies on Hydrodynamics of a Cyclone Separator Employed in a Circulating Fluidized Bed

K. M. Pandey, *Member, IACSIT* and M. Ray

***Abstract*—Hydrodynamics of a cyclone separator as element of a re-circulating fluidised bed circuit has been studied experimentally. The main efforts have been addressed the solids suspension density distribution along the cyclone axis at various solids circulation rates. In general, the suspension density decreases in the downward direction and becomes least in the cyclone conical section. At the cyclone exit it starts increasing and this trend persists in the dipleg of downcomer. The suspension density decreases as the bed inventory in inlet section of cyclone is increased and the solids circulation rate in the cyclone-downcomer branch of the fluidized bed circuit. It is to be noted that use of fluidized bed keeps the environment more clean.**

***Index Terms*—cyclone separator, suspension density, circulating fluidized bed, high solids inventory.**

I. Introduction

There is a vast literature available in context with fluidized bed operations but most of them are not given here in details. These are given in the references. Only few are discussed here who are relevant with the present study. The two laboratory-sized scale models have been designed to simulate the hydrodynamic behavior and study the effect of bed diameter on near-wall hydrodynamics by Noymer et al. [1] The influence of riser diameter on the axial and radial solids flux and flow development is studied in three riser circulating fluidized bed reactors of different diameters (76, 100 and 203 mm i.d. risers). A suction probe was used for the direct measurement, while the calculated local solids flux data obtained from solids velocity and concentration measured by two separate fiber optic probes were used to compare. Two shapes were found for the radial profile of the solids flux, a parabolic shape and a flat core shape. The shapes obtained could be predicted, based on the operating conditions, using a new concept of the effective solids saturation carrying capacity. The radial profile of solids flux is less uniform in a larger riser than in a smaller riser. Flow development is slower with the increase of riser diameter. The operating conditions were found to affect the solids flux in each reactor in the same general fashion: increasing gas velocity decreased the amount of down flow solids in the risers; increases in the solids circulation rate caused more solids to flow downwards. The following conclusions can be drawn from this study: Solids flux data determined by direct measurements are in good agreement with those calculated from the measured solids concentration and velocity. The shapes of the radial solids flux profiles in three risers were found to be either flat with decreasing annulus or parabolic, depending on the operating conditions. The shape of the radial solids flux profile could be predicted using the effective saturation capacity, above which a parabolic profile prevails due to the increased solids down flow near the wall. The radial profile of solids flux is less uniform in a larger riser than in a smaller riser. The amount of down flowing solids is highest in the lower sections of the riser, but then begins to decrease as the measuring point is moved away from the distributor. Increases in U_g (superficial gas velocity) cause the amount of down flowing solids at the wall to decrease, resulting in a more flat radial profile of the solids flux in the columns. Increasing G_s (overall solids circulating rate) increases the amount of down flowing solids, resulting in a more parabolic shape. The flow development is affected in the same manner in different diameter columns: increases in Ug or decreases in Gs reduce the length of development. Flow development is slower with the increase of riser diameter by Yan et al. [2]. The experimental work was carried out in a 12 MWth CFB boiler and in a cold CFR Three different distributions of the bubble flow in time and space, termed fluidization regimes, were identified in the cold CFB: the multiple bubble regime with many small bubbles evenly distributed in the bed; the single bubble regime, characterized by the presence of only one bubble at a time in the bed; and the exploding bubble regime with large, single, irregular voids stretching from the air distributor to the bed surface. These bubbling conditions were observed during variations in the gas velocity and the distributor pressure drop. A comparison with the 2-m2 cross-section CFB boiler showed that the boiler always operates in the single or in the exploding bubble regime, which indicates a bubble flow that is not continuous and not well distributed over the cross-section of the bed. Afsin Gungor worked on [3] a dynamic two dimensional model is developed considering the hydrodynamic behavior of CFB. In the modeling, the CFB riser is analyzed in two regions: The bottom zone in turbulent fluidization regime is modeled in detail as two-phase flow which is subdivided into a solid-free bubble phase and a solid-laden emulsion phase. In the upper zone core–annulus solids flow structure is established. Simulation model takes into account the axial and radial distribution of voidage, velocity and pressure drop for gas and solid phase, and solids volume fraction and particle size distribution for solid phase. The model results are compared

K.M.Pandey and M.Ray,Department of Mechanical Engineering, National Institute of Technology, Silchar, Assam, India, Paper submitted for review on 03-06-2010.

with and validated against atmospheric cold bed CFB units' experimental data given in the literature for axial and radial distribution of void fraction, solids volume fraction and particle velocity, total pressure drop along the bed height and radial solids flux. Ranges of experimental data used in comparisons are as follows: bed diameter from 0.05–0.418 m, bed height from 5–18 m, mean particle diameter from 67–520 μm, particle density from 1398 to 2620 kg/m3, mass fluxes from 21.3 to 300 kg/m2s and gas superficial velocities from 2.52–9.1 m/s. As a result of sensitivity analysis, the variation in mean particle diameter and superficial velocity, does affect the pressure especially in the core region and it does not affect considerably the pressure in the annulus region. Radial pressure profile is getting flatter in the core region as the mean particle diameter increases. Similar results can be obtained for lower superficial velocities. It has also been found that the contribution to the total pressure drop by gas and solids friction components is negligibly small when compared to the acceleration and solids hydrodynamic head components. At the bottom of the riser, in the core region the acceleration component of the pressure drop in total pressure drop changes from 0.65% to 0.28% from the riser center to the core–annulus interface, respectively; within the annulus region the acceleration component in total pressure drop changes from 0.22% to 0.11% radially from the core–annulus interface to the riser wall. Hideya Nakamura presented on [4] modeling of particle fluidization behaviors in a rotating fluidized bed (RFB) was conducted. The proposed numerical model was based on a DEM (Discrete Element Method)-CFD (Computational Fluid Dynamics) coupling model. Fluid motion was calculated two dimensionally by solving the local averaged basic equations. Particle motion was calculated two-dimensionally by the DEM. Calculation of fluid motion by the CFD and particle motion by the DEM was simultaneously conducted in the present model. Geldart group B particles (diameter and particle density were 0.5 mm and 918 kg/m^3, respectively) were used for both calculation and experiment. The calculated fluidization behaviors, such as periodic bubbling fluidization and transition of fluidization regime were in good agreement with the ones observed by a high-speed video camera. Comparison between the experimental results of both Umf and ΔPf and the estimated ones by our proposed model and other analytical models was conducted to evaluate the accuracy of our model. The calculated Umf and ΔPf showed a good agreement with the experimental results. Especially, the estimated ΔPf using our proposed model showed better agreement with the experimental results, while the other analytical models overestimated ΔPf by 30%.

S. Satish discussed on [5] The work involves experimentation on drying of solids in a continuous fluidized bed dryer covering different variables like bed temperature, gas flow rate, solids flow rate and initial moisture content of solids. The data are modeled using artificial neural networks. The results obtained from artificial neural networks are compared with those obtained using Tanks-in-series model. It was found that results obtained from ANN fit the experimental data more accurately compared to the RTD model with less percentage error. The resultant network as seen from the above results for the first data set is simple and its performance evaluated by the percentage error criterion is also satisfactory. On the other hand, introducing one more input variable makes the network more complex as seen from the results of the second set. It has been observed that back-propagation networks with two hidden layers outperform the single hidden layer networks when applied to prediction problems. M.J. Cocero worked on [6] Simulation results of supercritical fluidized bed by Fluent CFD software are presented. Simulation based on the multifluid Eulerian model incorporating the kinetic theory for solid particles and using the Gidaspowdrag function have been performed in supercritical carbon dioxide (sc-CO2) ambient in a range of pressure from 8 to 12MPa and temperatures from 30 to 50 ^{0}C. The fluid velocity has been varied from 1 to 6 times the minimum fluidization velocity (umf). The multifluid Eulerian model incorporated in the Fluent CFD software, incorporating the kinetic theory for solid particles and using the Gidaspow drag function has been successfully applied to model fluidization in sc-CO2 at high pressure (8–12MPa) and nearly ambient temperature (30–50 ^{0}C). The model predicts particulate fluidization in the studied operation conditions and the solid velocity vectors show an upward circulation of solids in the center of the bed (jet effect) and a downward movement next to the bed walls. F. Okasha and M. Miccio focused on [7] a simple mathematical model has been developed to simulate the wet jet in fluidized bed. The different stages involved inside the jet zone have been estimated and analyzed. The evaporation stage of traveling droplets through the jet flare has been treated. The rates of evaporation of each size at all positions along the jet flare have been estimated according to the velocities and surrounding conditions. The final droplet sizes have been determined. Moreover, the total evaporation rate from traveling droplets, before collision either with entrained sand particles or flare boundaries, has been estimated. The traveling droplets, partially evaporated, may collide and settle on entrained sand particles. The model predicts the settlement rates of liquid droplets on entrained sand particles. The total part evaporated from settled liquid has been estimated as well. The study has been applied to the pneumatic feeding of liquid fuel into fluidized bed combustors operating at 850 ^{0}C. The model has been utilized to predict the ratio of fuel vapor that releases inside the jet flare. The model results demonstrate that only very small droplets completely evaporate inside the flare. The liquid settling over the entrained sand particles plays an essential role in the fuel evaporation inside the flare. A mathematical model has been developed to simulate the wet jet zone. The model manages the different stages involved in the jet zone, especially, the settlement of liquid droplets on the entrained sand particles and estimates their characteristic rates. The model has no adjustable parameters. Aboozar Hadavand presented on [8] a mathematical model of the circulating fluidized bed combustion system based on mass and energy conservation equations was successfully extracted. Using these correlations, a state space dynamical model oriented to bed temperature has been obtained based on subspace method. Bed temperature, which influences boiler overall efficiency and the rate of pollutants emission, is one of the most significant parameters in the operation of these types of systems. Having dynamic and parametric uncertainties in the model, a robust control algorithm based on linear matrix inequalities (LMI) have been applied to control the bed temperature by input parameters, i.e. coal feed rate and fluidization velocity. Circulating fluidized bed (CFB) combustion systems are increasingly used as superior coal burning systems in power generation due to their higher efficiency and lower emissions. However, because of their non-linearity and complex behavior, it is difficult to build a comprehensive model that incorporates all the system dynamics. Circulating fluidized beds exhibit very complex hydrodynamics caused by interactions between the gas and the solid phases. The motion of gases and solids are driven by mechanisms that are difficult to identify and describe. A novel mathematical model of bed temperature in a circulating fluidized bed combustor was proposed. The mathematical model obtained which is established on mass and energy conservation equations, lets us incorporate all the process dynamics in the model. L.X. Kong and P.D. Hodgson worked on [9] To improve the understanding of the heat transfer mechanism and to find a reliable and simple heat-transfer model, the gas flow and heat transfer between fluidized beds and the surfaces of an immersed object is numerically simulated based on a double particle-layer and porous medium model. The velocity field and temperature distribution of the gas and particles are analysed during the heat transfer process. The double particle-layer and porous medium model has the ability to simulate the gas flow and the heat transfer near the surface of an immersed object in fluidized beds, and was successfully used in calculating the dynamic characteristics of the gas phase, the temperature change of particles and the radiative parameters of a particle group. The results provide sufficient information to improve the understanding of heat transfer processes near the immersed surface.

J.C.S.C. Bastos focused on [10] Radial solids velocity profiles were computed on seven axial levels in the riser of a high-flux circulating fluidized bed (HFCFB) using a two-phase 3-D computational fluid dynamics model. The computed solids velocities were compared with experimental data on a riser with an internal diameter of 76 mm and a height of 10 m, at a high solids flux of 300 $kg/m^2.s$ and a superficial velocity of 8 m/s. Several hundreds of experimental and numerical studies on CFBs have been carried out at low fluxes of less than 200 $kg/m^2.s$, whereas only a few limited useful studies have dealt with high solids flux. The k–e gas-particle turbulence model is compared against experimental radial solids velocities on seven axial levels in a 76 mm ID and at a height of 10 m in a high-flux circulating fluidized bed (HFCFB) at a high flux of 300 $kg/m^2.s$ and a superficial velocity of 8 m /s. The HFCFB riser is divided into three regions with a dense solids region in the bottom section (h = 0– 4 m), a developed flow (h = 4–8 m) and a dilute region in the upper part (h = 8–10 m).A three-dimensional model of a circulating fluidized bed gasifier was developed by [11] I. Petersen and J.Werther, which uses continuous radial profiles of velocities and solids hold-up with regard to the description of fluid mechanics. A complex reaction network of sewage sludge gasification is included in the model. In the simulation calculations the influence of the axial location and the number of feeding points was examined for gasifiers of different scales. It was found that due to the very fast decomposition of the volatiles and the high volatile content in the sewage sludge, lateral mixing of the gas around the feeding port is not complete, and plumes with high pyrolysis gas concentrations are formed. Due to the very fast release of the volatiles and the high volatile content in the sewage sludge, mixing of the gas around the feeding port is not complete, and plumes with high amounts of pyrolysis gas concentrations are formed. If the sewage sludge is fed below the level of the solids return into the bottom zone pyrolysis gas will be consumed by the fluidizing air and the risk of a hot spot near the fuel feed point arises. Vidyasagar Shilapuram worked on [12] Experiments were conducted in a liquid–solid circulating fluidized bed (LSCFB) to study the flow regimes, operational instability, critical transitional velocity to circulating fluidized bed (CFB) regime, solids holdup and solids circulation rate by three experimental methods. The results indicate that the operational instabilities such as arch formation, liquid–solid separator blockage and solids return pipe blockage were observed in two of these methods at large primary and auxiliary liquid velocities. The critical transitional velocity that demarcates the expanded bed from CFB regime was observed to be different by these three methods. The macroscopic flow properties (flow regimes, onset of average solids holdup, average solids holdup and solids circulation rate in the riser) are different by different methods of operation.

II. Problem Formulation

Previous research work has shown that the diameter of a Circulating Fluidized Bed (CFB) has a significant effect on the heat transfer rates to peripheral walls that is important for the application of CFBs as combustors and boilers. The phenomenon might be predicted since larger-diameter beds have lower perimeter to cross-sectional area ratios thus accumulating more particles on the circumference without changes in the internal recirculation of solids per unit area in the core. Therefore, the bed diameter affects both the hydrodynamics and the heat transfer that need laboratory tests prior increase in the device scale towards industrial-scale combustors.

A. *Objectivities and aim*

Following the problem formulation the objectivities of the work are:

1) Effect of the operating conditions on thesuspension density in the cyclone and downcomer.
2) Suspension density profiles along the cyclone axis.
3) Suspension density profiles along the downcomer equipped with either vertical or horizontal inserts.
4) Suspension density profiles along riser height and the effect of the operating conditions on them.

International Journal of Chemical Engineering and Applications, Vol. 1, No. 2, August 2010
ISSN: 2010-0221

III. Experimental

A. Experimental set-up

The experiments were performed with two laboratory-sized scale models were built and run at room temperature. The two units were of the same height and were run at the same operating conditions. The principle design is shown schematically in Fig. 1 and the principle operating conditions are summarized in Table I. The properties of the solids used are summarized in Table II.

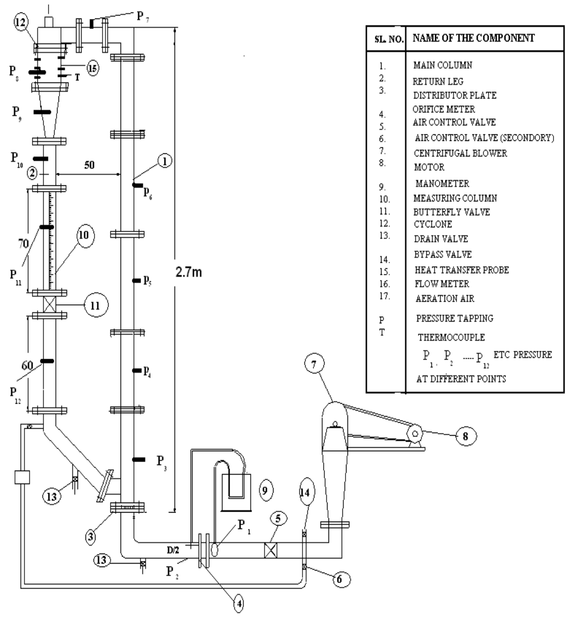


Fig. 1.1 schematic diagram of the experimental

TABLE I.The Basic Dimensions of the Test Units and Operating Conditions

Diameter	0.33 m and .05 m
Heights	14.3 m
Solids Concentration range	2.93–3.30 m/s
gas velocity ranges	2.93–3.30 m/s

TABLE II The Properties of the Solid

Bed Material	Sand
Mean particle size of sand:	370 μm
Fluidizing velocity	2.93–3.30 m/s
Solid circulation rate	Up to 14 kg/m^2 s
Bed inventories:	8 kg, 11 kg

The cyclone separator tested with the units is show schematically in Fig.1.2.

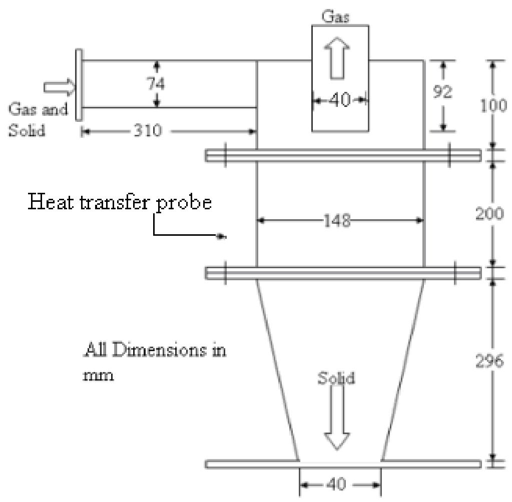


Fig. 1.2. The cyclone separator tested – schematically

IV. Measurement Techniques

A. Air flow and pressure drop

The rate of airflow through the bed was measured by using a standard orifice meter made to design with D and D/2 tapings. Equation is used to estimate the volume flow rate of air through the orifice meter.

Pressure tapings were provided at four different locations along the height of the riser in 0.6 m interval between two adjacent columns (Plexiglas) to determine the axial suspension density of the bed. Four number of pressure tapings were also provided along the axis of cyclone length to determine axial suspension density along the cyclone height. These pressure tapings were connected to water filled U-tube manometer. Fine wire mesh (B.S. 400) and cigarette filters were used at pressure tapping ends to minimize pressure fluctuations in the manometers.

B. Solids Circulation Rate

The butterfly valve located at the middle of the return leg was used to measure and controls the solid circulation rate in the CFB loop. Closing the valve and measuring the volume of solids collected above it over a certain period of time measured the solid circulation rate. At steady velocity the butterfly valve was closed sharply, and with the help of a stopwatch, the time is recorded to store a certain amount of solid above it.

V. Experimental Procedures

Prior to the starting of actual experiments, some trial runs were taken to have an idea about the control and measurement of operating parameters. A known quantity of sand was fed into the main column through the top of the cyclone. The bag filter then covered the outlet of the cyclone. The blower was started and air was allowed to flow through the riser column with the help of the air control valve. The airflow rate was slowly increased and bed starts expanding, and within a short period it attains the complete fluidization condition when the air velocity was further increased. And the terminal velocity of particles was exceeded; the entrainment of solid particles began, which was observed virtually through the Plexiglas columns. The desired airflow rate was then maintained by adjusting the flow through the orifice meter, which was ascertained from the pressure drop data across the orifice meter. The entrained solids were allowed to return to the main column by operating the aeration air circulation with butterfly valve in open position. Thus a continuous loop of emulsion was established. The following data were recorded during the experiments:

1) Manometer reading (connected across the orifice meter)
2) Manometer reading (connected along the bed height)
3) Manometer reading (connected along the cyclone height)
4) Data for solid circulation rate.

VI. Results And Discussion

A. *Variations of suspension density in the cyclone and the downcomer*

Variation of suspension density along the axial direction of cyclone separator as well as down comer has been presented in Fig.1.3(a-c). Three different superficial velocities, viz, 2.67 m/s, 2.9 m/s and 3.2 m/s were used for the experiments. Measurements were taken from the cyclone inlet (riser end) towards the downward direction (Fig. 1.3(b)).

The suspension density attains it maximum at the cyclone inlet, starts to decrease along the cyclone axis in the downward direction and becomes least at the cone region. Further, it increases again in the downward direction. The reason for this is the intensity of swirl flow which in the cone part is more intensive than in section close to the inlet. Hence, the lower swirl intensity, the higher particle deposition. At the cyclone exit the suspension density starts to increase and this trend persists in the dipleg of downcomer. The increase in the gas superficial velocity yields increased amount of solids into the cyclone, and consequently increased number of turns and particle contacts with the wall. The data in Fig. 1.3(c) correspond to two different inventories of 8kg and 11kg . For both cases the suspension density decreases with increase in the bed inventory only in inlet section of cyclone due the high solids loading in the riser. The later simply means that for same superficial velocity fewer amounts of solids per unit volume of the gas flow enter the cyclone. Further, in the downward direction of cyclone and in the downcomer, too the suspension density is comparatively higher for high inventories that are logical results since the particle concentration in the cyclone is straightforward proportional to the solids inventory.

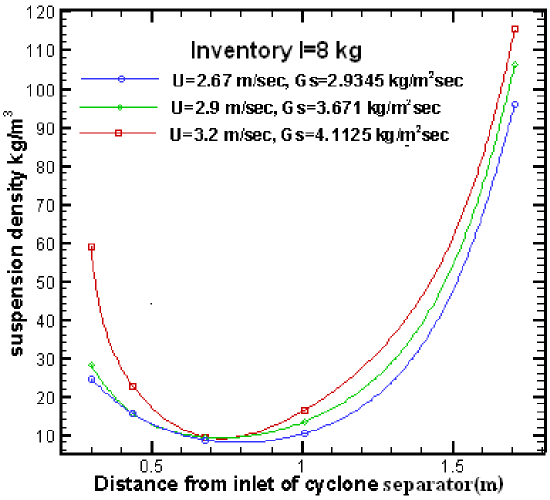


Fig. 1.3(a) Variation of suspension density along cyclone height and downcomer. Data: 8kg inventory of sand of grains of 275 μm (sieve size)

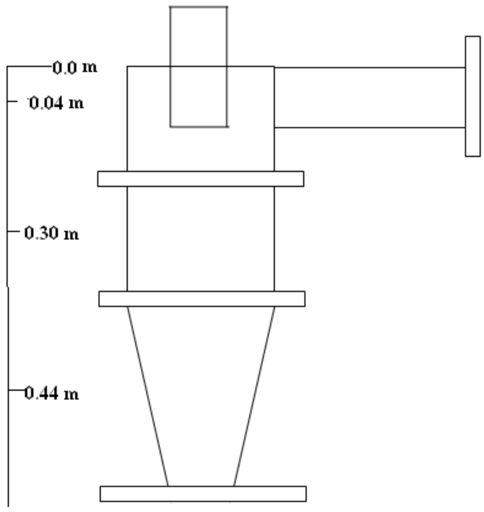


Fig. 1.3(b) Measuring points

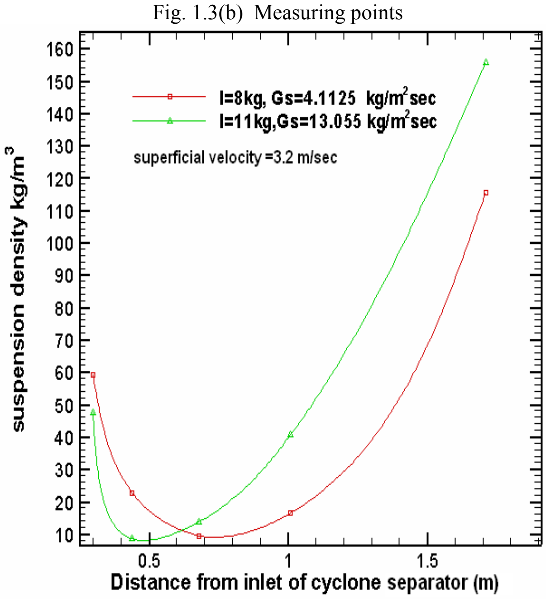


Fig. 1.3(c) Comparison of inventory for variation of suspension density along cyclone and downcomer. Superficial gas velocity 3.2 m/sec

ISSN: 2010-0221

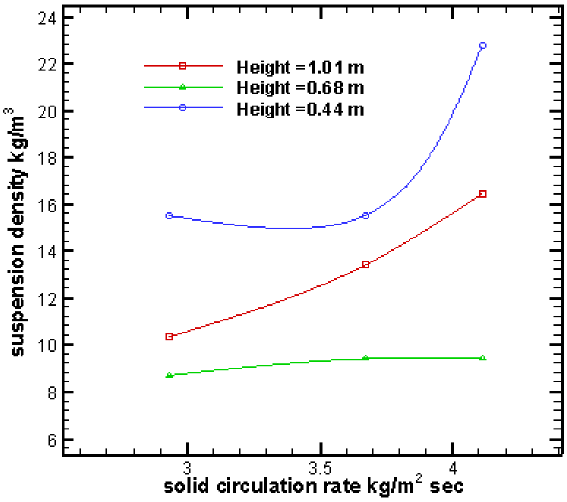


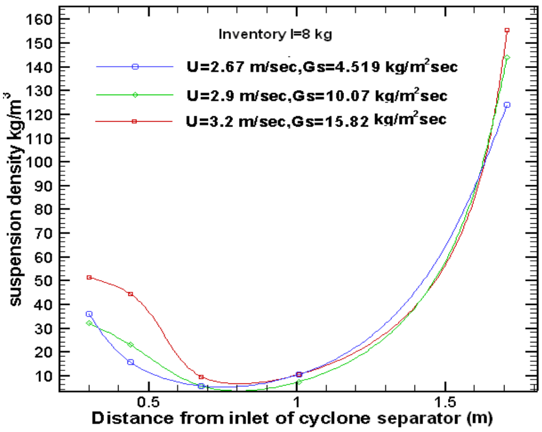
Fig. 1.3(d) Variation of suspension density with downcomer

Data concerning variations in the suspension density with variations in the solids circulation rate, measured at 3 different points (viz, 1.01m, 0.68m and 0.44m along the axis of cyclone to down comer) are presented in Fig. 1.3(d). These data simply reveal that the suspension density increases as the solids circulation rate is increased. Consequently, taking into account that that heat transfer coefficient increases linearly with suspension density more heat can be extracted from the solids in the cyclone inlet; the heat flux extracted in this section is greater than those removed in the downcomer and the cone section of the separator.

B. Axial suspension density profiles in the cyclone and the downcomer equipped with inserts

The axial suspension density profiles in the cyclone with different types of inserts are shown in Fig. 1.4 (a, b). The results are presented for 8 kg of bed inventory and three different superficial velocities. From these figures it is observed that the suspension density almost showing the same type of trend along the axis of cyclone and down comer when compared with cyclone which have no insert.

(a) with a vertical insert



(b) with a horizontal insert

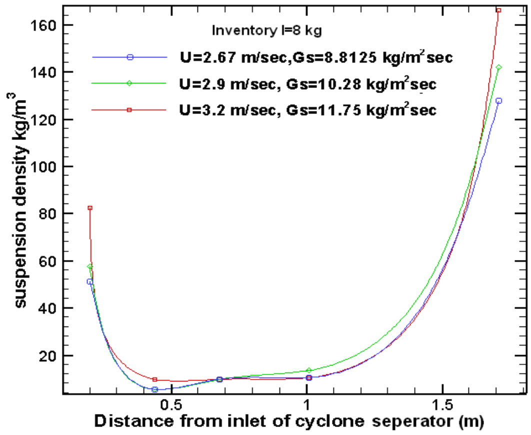
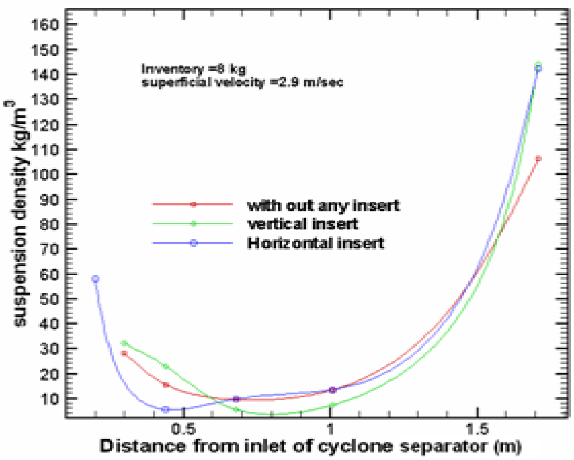


Fig.1.4 Effect of inserts on the axial suspension density profiles in the cyclone and the downcomer. 8kg solids inventory

The plots in Fig. 1.5 show the effect of the insert type on the suspension concentration profile inside the cyclone. These plots reveal that the suspension density in the separator inlet attained with the horizontal insert is higher than those with the vertical inset and the case of empty cyclone, respectively. These results imply increased solids concentration in the inlet section provoked by increased gas mixing due to the horizontal insert in comparison with that by the vertical counterpart; the obstruction to swirl in the cyclone core caused by the horizontal insert is stronger that that by the vertical one.

(a)



(b)

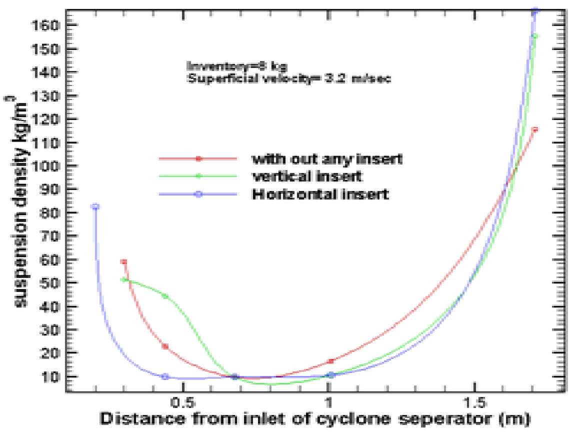


Fig. 1.5 Comparison of suspension density along cyclone with different type of insert. 8kg solids inventory.

The suspension density profiles along the cyclone axis down to the cone section indicate higher concentrations of the solids (almost twofold) promoted by the vertical insert then the horizontal one. This is due to solids segregation enhanced by the vertical tube arrangement. At the cyclone exit section there is no detectable effect of the type of the insert used and all the cases shown in Fig. 4c exhibit similar trends in the suspension density variations.

C. Axial suspension density profiles in the riser

It is generally recognized that the cross-sectional average suspension density is the most significant characteristic affecting the heat transfer between the bed and the furnace water wall. Commonly the heat transfer coefficients are presented as functions of the average suspension density. However the suspension density is not an independent parameter but varies with changes in many operating variables, among them: the type of particles used, the fluidizing velocity and the solids recycle rate, bed geometry and the bed hold up. Further, it may be relevant to point out that since the heat transfer is influenced by the gas and particle motion in the immediate vicinity of the heat transfer surface, the local suspension density is physically more meaningful than the cross-sectional average density. The results relevant to the axial suspension density profiles are shown in Fig. 1.6(a) at three different superficial velocities (viz, 2.67 m/sec, 2.9 m/sec, and 3.2 m/sec) .The plots clearly reveal that within an initial portion of about one meter of bed (measured from the bottom) the suspension density decreases drastically, i.e., the slopes of the curves in this initial section are higher than those observed further towards the riser exit.

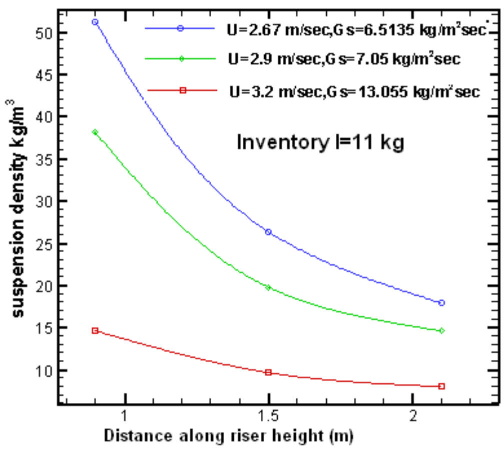


Fig 1.6(a) Axial suspension profile in the riser at various gas superficial velocities (i.e., solids circulation rates) 8 kg of bed inventory

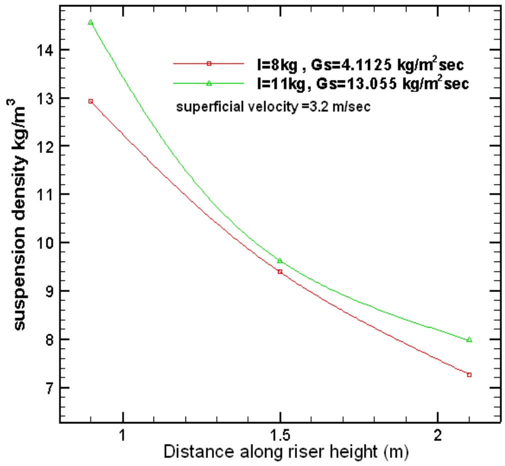


Fig.1.6(b) Effect of the solids inventory on the axial suspension profile in the riser at affixed gas superficial velocity (see the results in Fig. 1.6(b) too) 8 kg of bed inventory

The effect of the solids inventory on the axial suspension profile in the riser is shown in Fig. 1.6(b). Both figures show the effect of the solid circulation rate that may be co9ntroled by the solids inventory at a fixed gas flow rate (Fig. 1.6(b)) or vice versa (Fig. 1.6(a)).

VII. Conclusions

The experiments performed allow to draw some conclusions, among them:

- The suspension density is highest in the inlet of the cyclone separator. It decreases the downward parallel to the cyclone vertical axis and becomes least at the cone region.
- At the exit of the cyclone the separator suspension density starts increasing due to increase in particle concentration. The same trend persists in the dipleg of down comer.
- The suspension density increases with increase in the solids circulation rate at different heights of cyclone and down comer mainly affected by the solids inventory.
- Within an initial section of the riser (within 1 meter section of the bed measured from the bottom), the suspension density drops drastically and thereafter gradually decreased.

Acknowledgment

The author is highly obliged to Dr.P Mahanta, Professor and Head, Energy Center, IIT Guwahati, India for allowing to use the set up of fluidized bed by the concerned graduate student. The authors acknowledge the valuable suggestions from Prof. U.S.Dixit, Ex-Head, Department of Mechanical Engineering, I.I.T Guwahati, India from time to time during completion of this work. The authors acknowledge the financial help provided by AICTE from the project AICTE: 8023/RID/BOIII/NCP(21) 2007-2008.The Project id at IIT Guwahati is ME/P/USD/4.

References

[1] Peter D.Noymer, Matthew R. Hyre, Leon R. Glicksman, The effect of bed diameter on near-wall hydrodynamic in scale-model circulating

fluidized bed, International Journal of Heat and Mass Transfer, Vol. 43, pp.3641-3649, 2000.

[2] Aijie Yan, Jeff Ball, Jesse (Jingxu) Zhu, Scale-up effect of riser reactors (3) axial and radial solids flux distribution and flow development, Chemical Engineering Journal, Vol. 109,pp.97–106, 2005.

[3] Afsin Gungor, Hydrodynamic modeling of a circulating fluidized bed, Powder Technology 172 (2007) 1–13, Istanbul Technical University, Mechanical Engineering Faculty, Gumussuyu, 34437, Istanbul, Turkey.

[4] Hideya Nakamura, Numerical modeling of particle fluidization behavior in a rotating fluidized bed, Powder Technology 171 (2007) 106–117, Department of Chemical Engineering, Osaka Prefecture University, 1-1, Gakuen-cho, Sakai, Osaka 599-8531, Japan.

[5] S. Satish, Modeling of a continuous fluidized bed dryer using artificial neural networks, International Communications in Heat and Mass Transfer 32 (2005) 539–547, Department of Chemical Engineering, National Institute of Technology, Warangal, India.

[6] M.J. Cocero, Supercritical fluidized bed modeling, J. of Supercritical Fluids 50 (2009) 54–60, Department of Chemical Engineering & Environmental Technology, University of Valladolid, c/ Prado de la Magdalena s/n 47011 – Valladolid, Spain.

[7] F. Okasha and M. Miccio, Modeling of wet jet in fluidized bed, Chemical Engineering Science 61 (2006) 3079 – 3090, Department of Mechanical Engineering, Mansoura University, Mansoura, Egypt.

[8] Aboozar Hadavand, An innovative bed temperature-oriented modeling and robust control of a circulating fluidized bed combustor, Chemical Engineering Journal 140 (2008) 497–508, Department of Electrical Engineering, Iran University of Science and Technology, Elm-o-Sanat street, Narmak, Tehran 16846, Iran.

[9] L.X. Kong and P.D. Hodgson, Computational simulation of gas flow and heat transfer near an immersed object in fluidized beds, Advances in Engineering Software 38 (2007) 826–834, Centre for Material and Fibre Innovation, Deakin University, Geelong, Vic. 3217, Australia.

[10] J.C.S.C. Bastos, Modelling and simulation of a gas–solids dispersion flow in a high-flux circulating fluidized bed (HFCFB) riser, Catalysis Today 130 (2008) 462–470, State University of Campinas, Campinas, Sao Paulo, Brazil.

[11] I. Petersen, J.Werther, Three-dimensional modeling of a circulating fluidized bed gasifier for sewage sludge, Chemical Engineering Science 60 (2005) 4469–4484, Technical University Hamburg-Harburg, Denickestr. 15 D-21073 Hamburg, Germany.

[12] Vidyasagar Shilapuram, Comparison of macroscopic flow properties obtained by three methods of operation in a liquid–solid circulating fluidized bed, Chemical Engineering and Processing 48 (2009) 259–267, Department of Chemical Engineering, Indian Institute of Technology Madras, Chennai 600036, India.

[13] A.Svensson, F.Johnsson, B.Leckner, Fluidization regimes in non-slugging fluidized beds: the influence of pressure drop across the air distributor, Powder Technology, Vol. 86, 299-312,1996.

[14] H.T.Bi and J.R.Grace, Flow regime diagrams for gas-solid fluidization and upward transport, International Journal of Multiphase flow, Vol. 21, No.6, pp.1229- 1236, 1995

[15] E.H.van der Meer, R.B. Thorpe, J.F. Davidson, Flow patterns in the square cross-section of a circulating fluidized bed and the effect of riser exit design, Chemical Engineering Science, Vol.55, pp.4079-4099, 2000.

[16] G. R. Caicedo, M. G. Ruiz, J. J. P. Marques, J. G. Soler, Minimum fluidization velocities for gas-solid 2D beds, Chemical Engineering and Processing, Vol. 41, pp.761-764, 2002.

[17] Y. Molodtsof, Theoretical analysis of the flow regimes and their characteristics in vertically flowing gas-solid suspensions, Chemical Engineering Journal, Vol.96, pp. 133-143, 2003.

[18] B.Zethreus, A theoretical model for gas-particle contact efficiency in circulating fluid bed risers, Powder Technology, Vol. 88, pp.133-162, 1996.

[19] Yu.S. Teplitsky, V.A. Borodulya, E.F. Nogotov, Axial solids mixing in a circulating fluidized bed, International Journal of Heat and Mass Transfer, Vol. 46, pp. 4335–4343, 2003.

[20] Wennan Zhang, Filip Johnsson, Bo Leckner, Fluid-Dynamic Boundary layers in CFB boilers, Chemical engineering Science, Vol. 50, pp.201-210, 1995.

[21] Jeong-Hoo Choi, Tae-Woo Kim, Young-Sub Moon, Sang-Done Kim, Jae-Ek Son, Effect of temperature on slug properties in a gas fluidized bed, Powder Technology, Vol.131, pp.15-22, 2000.

[22] D.Gauthier, S.Zerguerras, G. Flamant, Influence of the particle size distribution of powder on the velocities of minimum and complete fluidization, Chemical Engineering Journal, Vol. 74, pp. 181-196, 1999.

[23] Sung Won Kim, Sang Done Kim, Effects of particle properties on solids recycle in loop-seal of a circulating fluidized bed, Powder Technology, Vol. 124, pp. 76– 84, 2002.

[24] Ulrike Lackermeier, Joachim Werther, Flow phenomena in the exit zone of a circulating fluidized bed, Chemical Engineering and Processing, Vol. 41, pp. 771- 783, 2002.

[25] J. Zhou, J. R. Grace, C. J. Lim and C. M. H. Brereton, Particle velocity profiles in a circulating fluidized bed riser of square cross-section, Chemical Engineering Science, Vol. 50, No. 2, pp. 237 – 244, 1995.

[26] Won Namkung, Sang Done Kim, Radial gas mixing in a circulating fluidized bed, Powder Technology, Vol. 113, pp. 23–29, 2000.

[27] Q. Guo, G.Yue, T. Suda, J. Sato, Flow characteristics in a bubbling fluidized bed at elevated temperature, Chemical Engineering and Processing, Vol. 42, pp.439-447, 2003.

[28] S.B.Schut, E.H. van der Meer, J.F. Davidson, R.B. Thrope, Gas-solids flow in the diffuser of a circulating fluidized bed riser, Powder Technology, Vol. 111, pp. 94-103, 2000.

[29] D. Sathiyamoothy and M. Horio, On the influence of aspect ratio and distributor in gas fluidized beds, Chemical Engineering Journal, Vol. 93, pp. 151-161, 2003.

[30] S. Roy, A. Kemoun, M. Al-Dahhan, M.P. Dudukovic, A method for estimating the solids circulation rate in a closed-loop circulating fluidized bed, Powder Technology, Vol. 121, pp. 213-222, 2001.

[31] M.Pozarnik, L.Skerget, Simulation of gas-solid flows by boundary domain integral method, Engineering Analysis with Boundary Elements, Vol.26, pp. 939-949, 2002.

[32] Sunun Limtrakul, Ativuth Chalermwattanatai, Kosol Unggurawirote, Yutaka Tsuji, Toshihiro Kawaguchi and Wiwut Tanthapanichakoon, Discrete particle simulation of solids motion in a gas-solid fluidized bed, Chemical Engineering Science, Vol. 58, Issues 3-6, Pages 915-921, 2003.

[33] Kefa Cen, Xiaodong Li, Yangxin Li,Jianhua Yan, Yueliang Shen, Shaorong Liang, Mingjiang Ni, Experimental study of a finned tines impact gas-solid separator for CFB boilers, Chemical Engineering Journal, Vol.66, pp.159-169, 1997.

[34] Minghui Zhang, Zhen Qian, Hao Yu, Fei Wei, The solid flow structure in a circulating fluidized bed riser/downer of 0.42 m diameter, Powder Technology, Vol. 129, pp.46-52, 2003.

[35] B.A.Andersson, Effect of bed particle size on heat transfer in circulating fluidized bed boilers, Powder Technology, Vol.17, pp.239-248, 1996.

[36] D.Xie, B.D.Bowen, J.R.Grace, C.J.Lim, Two-dimensional model of heat transfer in circulating fluidized beds. Part I Model development and validation, International Journal of Heat and Mass Transfer, 46, pp. 2179-2191, 2003.

[37] K.E.Wirth, Heat Transfer in Circulating fluidized beds, Chemical Engineering Science, Vol.50, pp.2137-2151, 1995.

[38] C.Breitholtz, B.Leckner, A.P.Baskakov, Wall average heat transfer in CFB boilers, Powder Technology, Vol.120, pp.41-48, 2001.

[39] A.V.S.S.K.S. Gupta, P.K.Nag, Bed-to-wall heat transfer behavior in a pressurized circulating fluidized bed, International Journal of Heat and Mass Transfer, 45, pp.3429-3436, 2002.

[40] S.Ouyang, X.G.Li, G.Davies, O.E.Potter, Heat transfer between a vertical tube bundle and fine particles in a CFB down comer with and without circulation, Chemical Engineering and Processing, Vol. 35, pp.21-27, 1996.

[41] Sung Won Kim, Jung Yeul Ahn, Sang Done Kim, Dong Hyun Lee, Heat transfer and bubble characteristics in a fluidized bed with immersed horizontal tube bundle, International Journal of Heat and Mass Transfer, vol.46, pp.399-409, 2003.1009201 1009134.

[42] A. Dutta and P. Basu, An Experimental investigation into the heat transfer on wing walls in a circulating fluidized bed boiler, International Journal of Heat and Mass Transfer, Vol. 45, pp. 4479-4491, 2002.

[43] P.K.Nag, M.Nawsher ali, P.Basu, A mathematical model for the prediction of heat transfer from finned surfaces in a circulating fluidized bed, International Journal of Heat and Mass Transfer, Vol.35, No.9, p.1675-1683,1995.

[44] M.R.Golriz, J.R.Grace, Augmentation of heat transfer by deflectors on circulating fluidized bed membrane walls, International Journal of Heat and Mass Transfer, Vol.45, pp.1149-1154, 2002.

[45] Josmar Davilson Pagliuso , Geraldo Lombardi , Leonardo Goldstein Jr., Experiments on the local heat transfer characteristics of a circulating fluidized bed, Experimental Thermal and Fluid Science, Vol. 20, pp. 170-179, 2000.

[46] R. Sundaresan, Ajit Kumar Kolar, Core heat transfer studies in a circulating fluidized bed, Powder Technology, Vol. 124, pp. 138–151, 2002. [38] C. Lockhart, J. Zhu, C. M. H. Brereton, C. J. Lim and J. R. Grace, Local heat transfer, solids concentration and erosion around membrane tubes in a cold model circulating fluidized bed, International Journal of Heat and Mass Transfer, Vol. 38, No.13, pp. 2403-2410, 1995.
[47] W. B. Fox, N. S. Grewal, and D. A. Moen, Wall-to-bed heat transfer in circulating fluidized beds, International Journal of Heat and Mass transfer, Vol. 26, No. 4, pp. 499-508, 1999.
[48] Sivkumar Kulasekaran, Temi M. Linjewile,& Pradeep K. Agarwal, "Mathematical modeling of fluidized bed combustion 3. Simultaneous combustion of char and combustable gases" Fuel 78(1999) 403-417.
[49] Raj A. Srinivasan, Suresh Sriramulu, Sivakumar Kulasekaran and Pradip K. Agarwal, "Mathematical modeling of fluidized bed combustion-2: combustion of gases" Fuel Vol. 77, No.9/10, pp.1033-1049, 1998.
[50] Suresh Sriramulu, Samir Sane, Pradip K. Agarwal and Tarek Mathews, "Mathematical modeling of fluidized bed combustion-1.Combustion of carbon in bubbling beds" Fuel Vol. 75, No. 12, pp. 1351-1362,1996 .
[51] Z. Chen, Pradip K. Agarwal and J.B. Agnew, "Steam drying coal. Part 2. Modeling the operation of a fluidized bed drying unit" Fuel 80 (200) 209 -223.

Dr. K. M. Pandey did his PhD in Mechanical Engineering in 1994 from IIT Kanpur. He has published and presented 150 papers in International & National Conferences and Journals. Currently he is working as Professor of the Mechanical Engineering Department, National Institute of Technology, Silchar, Assam, India..He also served the department in the capacity of head from July 07 to July 13, 2010. He has also worked as faculty consultant in Colombo Plan Staff College, Manila, Philippines as seconded faculty from Government of India. His research interest areas are the following; Combustion, High speed flows, Technical Education, fuzzy logic and neural network , heat transfer, internal combustion engines, human resource management, gas dynamics and numerical simulations in CFD from commercial softwares.