EXPERIMENTAL AND COMPUTATIONAL STUDIES OF THE HYDRODYNAMICS OF THE RISER OF A CIRCULATING FLUIDIZED BED

A. Hussain, F.N. Ani, A.N. Darus
Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310, Skudai, Johor, Malaysia

A. Mustafa
Faculty of Chemical and Natural Resources, Universiti Teknologi Malaysia, 81310, Skudai, Johor, Malaysia

Correspondence Email: ahmad@siswa.utm.my

ABSTRACT

The hydrodynamics of a circulating fluidized bed (CFB) risers is highly complex and is strongly influenced by the distribution of particles, which is governed by the amount of particles, size, form (e.g. spherical or elliptic), density, etc. The CFB riser column has 265 mm (width), 72 mm (depth) cross-section (rectangle) and 2.7 m height. The riser is made up of interchangeable Plexiglas columns. In this study the influence of the amount of particles on the flow pattern in the CFB system is investigated. The particle loading is increased from a dilute to a dense and gas-particle flow is analyzed.

Simulations were done using FLUENT 6.1, a CFD package by Fluent Inc. Palm shell particles and air were used as the solid and gas phases, respectively. The 2D simulations were done using the geometrical configuration of a CFB test rig at the Universiti Teknologi Malaysia (UTM).

The result discusses the variation of velocity contours, along the riser column and in the riser exit geometry. The effect is significant in the upper region of the riser column and the velocity contours are also influenced by the exit geometry. Simulations results predict that the riser exit cause an upstream exit region of increased solid volume fraction. Experimental and computational results are matched to reasonable agreement. Experimental findings have also helped to refine the numerical modelling of multiphase CFB system.

Keywords: CFD, FLUENT, hydrodynamics, multiphase flow, riser exit, velocity contours

1. INTRODUCTION

Circulating fluidized beds have been studied intensively during the past two decades in order to continuously improve industrial processes such as CFB combustion and fluid-catalytic cracking. (FCC). FCC units are used in most refineries all over the world to convert high molecular weight gas oils or residuum stocks into lighter hydrocarbon products in a riser reactor within a few seconds [1]. The circulating fluidised bed (CFB) is an advantageous alternative for combustion of solid fuels. This is because the fuel flexibility is high, and it is possible to control the combustion temperature. Due to the significance of CFB in industry and their complex fluid dynamics, more and more research papers on CFBs are being reported in the literature [2].

The performance of a CFB boiler is influenced by the mixing of gas and particles. A high mixing rate contributes to an effective distribution of reactants, whereas an insufficient mixing can lead to hydrocarbon and CO-emissions. Therefore, an adequate understanding of the mixing behaviour is important to ensure a high combustion efficiency and emission control. Knowledge on the mixing characteristics is also useful for validation of computer simulations of CFB risers.

CFB technology is now finding applications in biomass combustion. Biomass is a renewable resource with almost zero or very low net CO_2 emission since carbon and energy are fixed during the biomass growth. Compared with the other renewable energy resources, biomass is abundant in annual production, up to 2740 Quads, with geographically widespread distribution in the world [3].

The ability of CFBC systems to operate with a wide range of fuel types has been confirmed through extensive operational experience. The high degree of fuel flexibility that characterizes many designs of CFBC often allows a plant operator to select fuels on the basis of what may be currently available at an economic price and, where appropriate, produce a fuel blend that combines several such elements [4].

2. HYDRODYNAMICS OF A CFB

Hydrodynamics play s a crucial role in defining the performance of circulating fluidized bed (CFB) reactors. This awareness promoted a remarkable number of studies, which appeared in the scientific literature in the past 15 years. It led to satisfactory understanding of several aspects of gas – solids suspension behaviour in risers of different sizes and shapes, operated under a variety of conditions. There are, however, some important areas where available information is limited, and the poor degree of knowledge hinders the design and/or the operation of the industrial CFB reactors [5].

It is necessary to understand the hydrodynamic characteristics of the inlet region of risers where strong momentum interaction between gas and solids exists, and the first acceleration section where initial gas-solid profiles of velocity and solids concentration begin to form. The solids holdup, the length of the acceleration section, hydrodynamic mixing and transfer phenomena in risers are all influenced heavily by the riser inlet design. Factors that also affect riser bottom operations include the condition and rate of entering solids, the arrangement of main air and secondary air inlets at the riser bottom and so on [6].

A circulating fluidised bed (CFB) for industrial use often has a vertical riser of square cross-section to convey the upward co-current flow of gas and solids. A square or rectangular cross-section is common for CFB combustors (CFBCs). These slab-sided risers are adopted for ease of fabrication and the cross-section may be very large, e.g. 8 x 8 m or larger. By contrast, basic scientific work on CFBs is usually done on risers of circular cross-section, to simplify the flow pattern; a typical riser diameter for such studies is in the range 0.1-0.2 m [7]. At UTM, a rectangular crosssection test rig (72 mm x 265 mm) having a height of 2.3 m has been developed. It is useful in studying the end effects in rectangular crosssection geometry.

The gas–solids flows in the dense pneumatic conveying exhibit many interesting features related to their three-dimensional and unsteady nature. The dominant characteristic in this type of flows is the appearance of discrete flow instabilities commonly known as 'slugs' and 'plugs'. High-speed camera measurements can be processed and presented in several different ways to enable a better understanding of the flow behaviour, body shape type information, internal structure of the flow instabilities, their propagation velocity and frequency characteristics [8].

Another important aspect of the CFB is the introduction of secondary air injection, which is used, in industrial applications. The practice of secondary injection is particularly important in CFB boilers where there is need to limit the NOx formation. The total combustion air is split into primary stream, fed through the bottom distributor and secondary stream, injected at a certain level. However most of the investigations on CFB hydrodynamics have been carried out by injecting the fluidizing gas only at the bottom of the riser, so there is a need to analyze the effect of secondary air injection in detail, which has been accounted for in this study.

3. FLUIDIZATION REGIMES

In recent years, although an increasing number of literature have been devoted to circulating fluidized bed (CFB), the prediction of velocities over which different fluidization regimes exists is still difficult. Understanding of the flow regimes in CFB risers is the key to the successful design and scale up of CFB systems [9].

According to Kunni and Levenspiel [10], for given particles and given superficial gas velocity through the bed, we need to find what contacting regimes is involved whether it is packed bed, bubbling bed or circulating bed, with its sub regimes turbulent fluidized, fast fluidized or pneumatic transport. To define this, we must evaluate the dimensionless measure of particles size and gas velocity. These are defined as:

$$d^* = D_p\left[\frac{g\rho_g(\rho_p - \rho_g)}{\mu^2}\right]^{\frac{1}{3}} \tag{1}$$

$$u^* = u\left[\frac{\rho_g^2}{\mu g(\rho_p - \rho_g)}\right]^{\frac{1}{3}} \tag{2}$$

The solids will be suspended when pressure drop exceeds the weight of solids. This happens when the gas velocity exceeds the minimum fluidization velocity. This velocity is given in the dimensionless form as:

$$u_{min} = \frac{150(1-\varepsilon_{mf})u^*_{mf}}{\varepsilon^3_{mf}} + \frac{1.75(u^*_{mf})^2 d^*_p}{\varepsilon^3_{mf}} \tag{3}$$

Individual particles are blown out of the bed when the gas velocity exceeds the terminal velocity, u_t. Kunni and Levenspiel [11], gave this velocity for spherical particles as:

$$u_t = \left[\frac{18}{(d^*_p)} + \frac{0.591}{(\sqrt{d^*_p})}\right]^{-1} \tag{4}$$

and for the non spherical shaped particles Φ_s:

$$u^*_t = \left[\frac{18}{(d^*_p)^2} + \frac{2.335 - 1.744\Phi_s}{(\sqrt{d^*_p})}\right]^{-1} \tag{5}$$

As the gas velocity of the bubbling fluidized bed (BFB) increases, the bubbling action becomes very violent, bubbles coalesce and become very large and finally expand to form a core space in the dense region of the vessel. At the same time the cloud and emulsion merge and retreat to the walls of the vessel. In this state we have a fast fluidized contactor (FF). Between the BFB and FF regimes we have a difficult to describe turbulent bed (TB). At even higher velocities the wall region thins, dissolves, as the vessel enters the pneumatic conveying regimes (PC). This is being illustrated in Figure. 1.

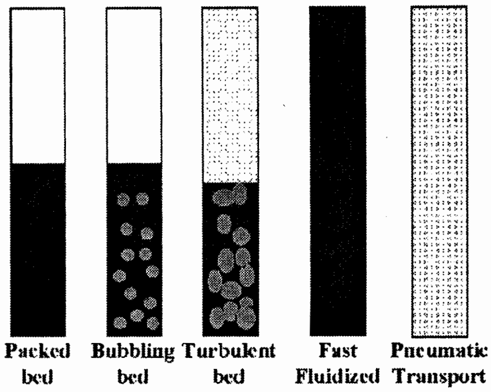

Figure 1: Fluidization regimes in vertical risers

4. CFB TEST RIG at UTM

In order to study fluidization behaviour of a CFB, a lab-scale true CFB test rig was built to fluidize various solids e.g. sand, palm shell waste, rice husk etc. The scale of the design is typical of other model CFBs that have appeared in the literature. The schematic diagram of the CFB is shown in Figure 2.

The system consists of an air supply device, a distributor of stainless steel, a fast column of Plexiglas, primary and secondary cyclones of stainless steel and a solid feeding system.

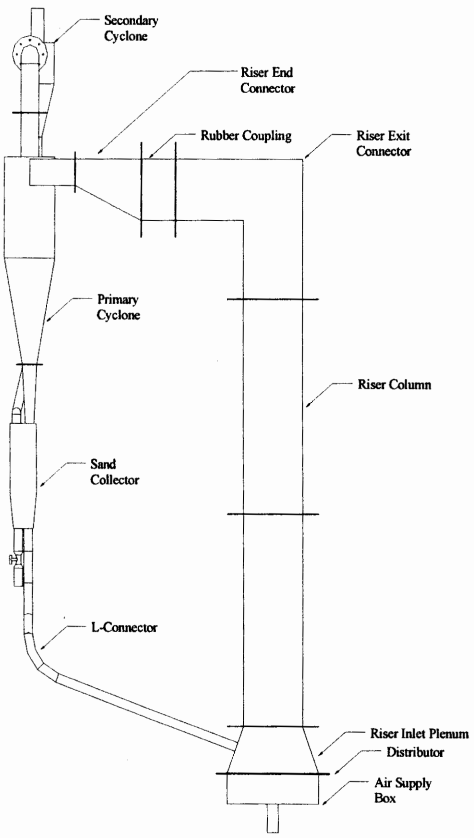

Figure 2: CFB test rig at Universiti Teknologi Malaysia.

5. EXPERIMENTAL METHOD

The pressure measurement for the CFB loop was done using the multi tube manometer. All the pressure taps fed to multi tube manometer tubes. The pressure tap through the riser wall had a copper tube inserted into it. Typical arrangement for the pressure measuring system is shown in Figure 3.

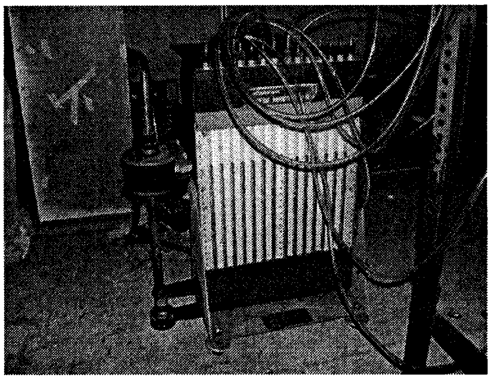

Figure 3: Pressure measurement using multitube manometer

The axial profile for particle distribution in a CFB riser is typically composed of five sections: the acceleration, developed bottom-dense, transition, top-dilute and exit sections. Usually, the acceleration and developed bottom-dense sections are together termed the bottom-dense (lower dense) section.

In the studies of the axial particle distribution, many authors [12, 13,14, 15,16,17,18] have assumed that the pressure gradient at an axial position is proportional to the amount of solid at that position.

6. RESULTS AND DISCUSSION

Commonly used names for the region of large solids volume fraction are dense region, bottom bed and choked bed. The transition zone is frequently referred to as the acceleration, splash or developing flow region. The region of low solids volume fraction is known as the dilute, transport or fully developed flow region. It is being shown in Figure 4.

The experimental observation had led to the understanding that the riser flow is generally characterised by: (i) an nonlinear, axial solids volume fraction profile, (ii) upward solids motion in the core of the riser and downward motion along the walls (Core/Annulus, or C/A flow), and (iii) a tendency of solids to form clusters.

Park [19] suggests a new hydrodynamic mode; to represent the gas flow in the dense phase of fluidized Group A powders. The model views that the particles form clusters under the influence of inter-particle forces, giving rise to the formation of a heterogeneous void structure consisting of clusters of particles and interstitial cavities.

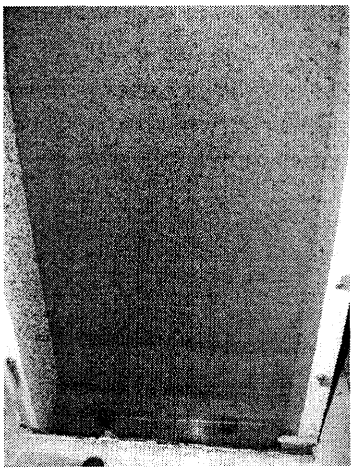

Figure 4: Fluidization behaviour for 600 μm palm shell waste particles.

The experimental results, suggest that riser flow is characterised by continuous formation and disintegration of clusters, and that clusters come in a variety of shapes, sizes, velocities and solids volume fractions. In the acceleration and dilute region, the number and size of clusters appear to increase towards the wall and be maximum near corners. Core clusters may move upwards or downwards, whereas wall clusters generally move downwards. The number of clusters decreases with increasing elevation, especially near the centre of the cross-section. The cluster size near the wall decreases with increasing elevation. Fluidization behaviour for the 1180 μm palm shell powder is being shown in Figure 5.

Figure 5: Fluidization behaviour for 1180 μm palm shell waste particles

Visual observations by of the flow in the riser exit, suggest dunes of significant size in the riser exit connector. It appears that solids in the horizontal connector may settle under gravity, which means that the remainder of the suspension is accelerated. Acceleration leads to a higher solids velocity in the cyclone and improves its efficiency Similar findings have also been reported by [20].

Although the distinction between `solids turbulence' and `gas turbulence' may be primarily of interest to simulations, it seems reasonable to conclude that full understanding of fast fluidisation cannot be obtained without considering turbulence. It seems likely that an interaction between collisions and turbulence is responsible for the macroscopic flow pattern.

An interaction between turbulence and collisions seems consistent with observations of a continuous formation and disintegration of clusters. Clusters may uphold a degree of stability against turbulence and collisions, because they experience less drag than individual particles. In addition, reduced drag increases the slip velocity and allows clusters to collect particles and grow in size.

Owing to their high density, solids in the core region of a riser exit may slip either to the outside or to the inside of the exit, dependent on the relative magnitudes of their inertia and the acceleration due to gravity g as shown in Figure 6. The ratio of solids inertia to gravity may be represented by the following Froude number:

$$Fr_R = \frac{\bar{u}_{st}^2}{gR} \tag{6}$$

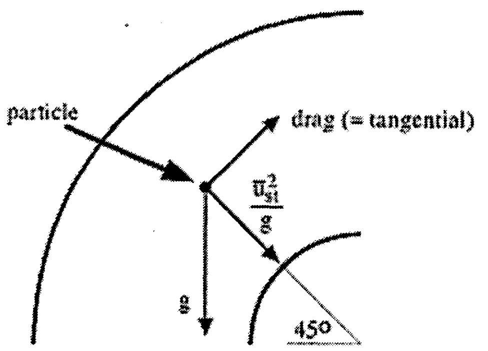

Figure 6: Contours of velocity profile using FLUENT 6.1

A high momentum suspension in the core of a riser exit may invoke similar lateral patterns. Simulation studies from FLUENT 6.1 shows secondary flow patterns in the riser exit. This is in agreement to the visual observations made during the experiments. It appears that velocity gradients are large near corners and in the middle of the inner and outer wall. Due to their high inertia, solids may accumulate in these areas. The velocity contour is being represented in Figure 7.

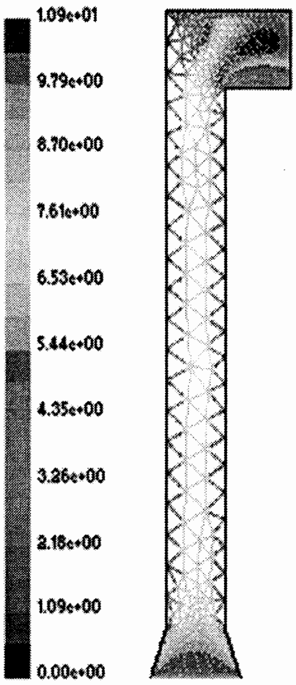

Figure 7: Contours of velocity profile using FLUENT 6.1

7. CONCLUSION

The axial particle distribution in circulating fluidized beds is important. It determines the pressure drop along the CFB. The CFB can be divided into two zones, a dense zone at the bottom and a dilute zone at the top of the riser.

It is being observed that the solids are accelerating and have a velocity much less their superficial velocity. Relatively, the solids density is higher in the top section. With an increase in height, the solid flow becomes fully developed and moves up with a constant velocity.

The solids hold-up can increase with elevation towards the top of the riser and core annular flow can be asymmetric in this region. Solids hold-up and asymmetry of flow are a function of the geometry of the riser geometry.

An interaction between turbulence and collisions can account for cluster formation and disintegration, and a lateral equilibrium with more solids in the annulus than in the core. The solids mean size in the downward flow was much bigger than the average size in the total CFB loop.

ACKNOWLEDGEMENT

The authors would like to thank Malaysian Government Commonwealth Secretariat, Public Service Department Malaysia, Putrajaya for funding the PhD program. Thanks are also due to Universiti Teknologi Malaysia for providing financial supports for developing and building the CFB test facility in the Department of Mechanical Engineering, UTM, Skudai, Johor, Malaysia.

NOMENCLATURE

D_p = particles diameter, m
g = gravitational acceleration, m/s^2
u^* = superficial velocity m/s
u = velocity m/s
u^*_{mf} = Minimum fluidization velocity
d* = dimensionless particle size
u_t = terminal velocity m/s

GREEK LETTERS

ρ_g = gas density kg/m^3
ρ_p = particles density kg/m^3
μ = viscosity of gas kg/m.s
ε_{mf} = minimum porosity
Φ_s = Sphericity of particles

REFERENCES

[1] Yan A., Parssinen, Zhu J. X. Flow properties in the entrance and exit regions of a high flux circulating fluidized riser. *Powder Technology* 131, 256-263, 2003.

[2] Cheng Y., Wei F., Yang G., Jin Y. Powder Technology 98, 151-156, 1998.

[3] Chen G., Andries J. and H. Spliethoff. Biomass conversion into fuel gas using circulating fluidised bed technology: the concept improvement and modeling discussions. Renewable Energy 28, 2003.

[4] Minchener A. J. Fluidized bed combustion systems for power generation and other industrial applications. Proceedings of the Institution of Mechanical Engineers, 2003.

[5] Mastellone, M.L. and U. Arena. The effect of particle size and density on solids distribution along the riser of a circulating fluidized bed. Chemical Engineering Science 54, 1999.

[6] Sterneus, J., Johnson F., Leckner B. Gas mixing in circulating fluidised bed risers. Chemical Engineering Science 55, 129-148, 2000.

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

[8] Jaworski A. J. and Dyakowski T. Investigations of flow instabilities within the dense pneumatic conveying system. Powder Technology 125, 279-291, 2002.

[9] Monazam E. R. and L. J. Shadle. A transient method for characterizing flow regimes in a circulating fluid bed. Powder Technology 139, 2004.

[10] Kunii, D., and O. Livenspiel. Circulating fluidized reactors. Chemical Engineering Science 52, 1997.

[11] Kunii, D. and O. Livenspiel. Fluidization Engineering, 1991.

[12] Xu, G. and Shiqiu, G. Necessary parameters for specifying the hydrodynamics of circulating fluidized bed risers – a review and reiteration. Powder Technology 137, 63-76, 2003.

[13] Davidson, J.F. Circulating Fluidized bed hydrodynamics. Chemical Engineering Science 113, 249-260, 2000.

[14] Contractor, R., Dry R.J., White C., Mao Q.M., Konstantinidis S. and Potter O. E. Circulating fluidized beds-diameter, solid hold up, axial gas mixing and contact efficiency. Powder Technology 111: 132-144, 2000.

[15] Mandal, S., D. K. Acharjee and P. S. Gupta. Distribution of Axial Voidage in the riser of a Circulating Bed. Indian Journal of Chemical Technology, Vol. 2, January 1995.

[16] Weinstein, H., L. Chen, and A. Kostazos. The Radial Pressure Gradient in a Fluidized Beds Risers. Fluidization and Fluid-Particle Systems, AIChE Annual Meeting, 1995.

[17] Van der Ham, A. G. J., W. Prins and W. P. M. van Swaaij. Hydrodynamics of a Pilot Plant Scale Regularly Packed Circulating Fluidized Bed. Fluid-Particle Processes: Fundamentals and Applications, AIChE Symposium Series, 1993.

[18] Yerushalmi, J. An Overview of Commercial Circulating Fluidized Bed Boilers. Circulating Fluidized Bed Technology II, 1985.

[19] Park, J.Y. The clustered dense phase models for group A fluidization: Dense phase hydrodynamics. Chemical Engineering Science, Vol. 58: 193-202, 2003.

[20] Muschelknautz, E, U. Muschelknautz. Special design of short entrance ducts to recirculating cyclones. Proc. 5th Int. Conf. Circ. Fl. Bed, preprint volume, Eq6 1 –6, 1996.