Simulation of Gas-Solid Flow Behaviour in the Riser Section of a Circulating Fluidized Bed Using Computational Fluid Dynamics Software

Ahmad Hussain[1,*], Farid Nasir Ani[1], Amer Nordin Darus[1], W. B. Wan Nik[2], Azeman Mustafa[3] and Arshad Salema[3]

[1] Faculty of Mechanical Engineering, Universiti Teknologi Malaysia,
81310 UTM Skudai, Johor, Malaysia.
[2]Faculty of Science and Technology, Kolej Universiti Sains dan Teknologi Malaysia,
Mengabang Telipot, 21030 Kuala Terengganu, Malaysia.
[3]Faculty of Chemical and Natural Resources Engineering, Universiti Teknologi Malaysia,
Skudai, 81310, Johor, Malaysia

Abstract: The concept of circulating fluidized bed (CFB) is widely used in industry for catalytic reactions, power production and calcinations of rocks. The gas particle flow inside a CFB unit is very complex. A clear understanding about the hydrodynamics of a CFB can improve the design significantly. CFD modeling of gas particle flow may lead to improved understanding of such system and reduce the need for experimental analysis. A two dimensional and isothermal flow was simulated for the continuous phase (air) and the dispersed phase (solid particles). Conservation equations of mass and momentum for each phase were solved using the finite volume numerical technique. A numerical parametric study was performed on the influence of various physical aspects over the hydrodynamics of gas-solid two-phase flow in a riser. The geometrical configurations of the riser were adopted from of an experimental CFB test rig at the Universiti Teknologi Malaysia, which is still in commissioning phase. A Eulerian continuum formulation was applied to both phases. A two-dimensional computational fluid dynamics (CFD) model of gas-particle flow in the CFB has been established using the code FLUENT. The computational model was used to simulate the riser over a wide range of operating and design parameters. In addition, several numerical experiments were carried out to understand the influence of riser end effects, particle size, gas solid velocity and solid volume fraction on the simulated flow characteristics. Gas and particle flow profiles were obtained for velocity, volume fraction and turbulence parameters for each phase. The computational results were typical of the experimental results reported in literature. Our computational results showed that the inlet and outlet designs have significant effects on the overall gas and solid patterns and cluster formations in the riser. These results were found to be useful in further development of modeling of gas solid flow in the riser.

Keywords: CFD, Eulerian, FLUENT, hydrodynamics, riser, turbulence model

1. INTRODUCTION

Circulating fluidized beds (CFBs) 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 fluidized 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.

The fundamental problem encountered in modeling hydrodynamics of fluidized bed is the motion of the two phases of which the interface is unknown and transient, and the interaction is understood only for a limited range of conditions [3]. The first intuition in resolving the two-phase mixture is to treat each phase by standard continuum mechanics with boundary and jump conditions to solve the governing equations at the interfaces [4]. However, the mathematical complexities of the non-linearity of the equations and in defining the interpenetrating and moving phase boundaries make numerical solutions very difficult.

Despite the modeling challenges, application of CFD to model fluidized bed hydrodynamics continues to develop, as it has many advantages including design optimization and scale-up of such systems. Some of the correlations used in the models, however, remain to be empirical or semi-empirical. As a result, the model and its parameters must be validated against experimental measurements obtained at similar scale and configurations. In this study, the hydrodynamic behaviour of a circulating fluidized bed reactor was investigated. CFD simulation were done using commercial CFD software, Fluent. Various operating and boundary conditions were applied for the calculation of velocity profiles [5,6].

Corresponding author: ahmad@siswa.utm.my

2. MATERIALS AND METHODS

Fig.1 shows the schematics of the CFB used for simulation. The system consists of an air supply device (blower), a distributor of stainless steel, a fast column of Plexiglas and primary and secondary cyclones of steel and a solid feeding system. The riser and its exit are made of Plexiglas to visualize the flow behaviour and to perform image analysis.

The simulation work described here was done on a riser of rectangular cross-section of 265 (width) x 72 (depth) x 2649 (height) mm^3.

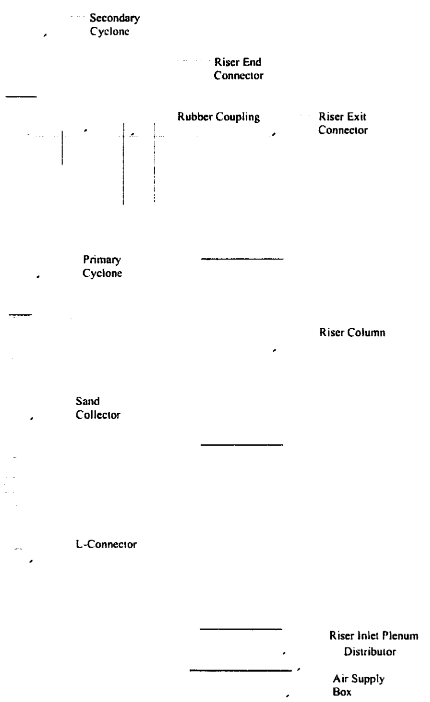

Fig. 1 Circulating fluidized test rig at Universiti Teknologi Malaysia

The parameters used in the simulation work are being summarized in Table 1.

Table 1 Parameters used in simulation work

Parameter	Range of values
Riser dimensions	2649 (L) X 265 (W) mm X 72 (T) mm
Gas velocity	5 m/s
Particle velocity	2 m/s
Properties of air	Density: 1.225 kg m^{-3} Viscosity: 1.79 x 10^{-5} kg/m.s
Properties of particle (sand)	Density: 2500 kg m^{-3} Diameter: 100X 10^{-6} m
Height of the sand inlet from distributor	200 mm
Exit Geometries Simulated	Right angle exit, right angle exit with baffle and blind T exit.
Volume fraction of sand	0.03

The operating parameters were chosen to so give dynamical similarity with large circulating fluidized bed combustors (CFBCs). Simulations were done using FLUENT, a computational fluid dynamics (CFD) package by Fluent Inc. Sand particles and air were used as the solid and gas phases, respectively. The meshing was done using Gambit.

3. EULERIAN MULTIPHASE MIXTURE MODEL

The FLUENT modeling is based on the three-dimensional conversation equations for mass, momentum and energy. The differential equations are discretized by the Finite Volume Method and are solved by the SIMPLE algorithm. As a turbulence model, the k-ε was employed; this consists of two transport equations for the turbulent kinetic energy and its dissipation rate. The FLUENT code utilizes an unstructured non-uniform mesh, on which the conservation equations for mass, momentum and energy are discretized. The k-ε model describes the turbulent kinetic energy and its dissipation rate and thus compromises between resolution of turbulent quantities and computational time.

Table 2 List of FLUENT Models used in simulation

Model	Settings
Space	2D
Time	Steady
Viscous	Standard k-epsilon turbulence model
Wall Treatment	Standard Wall Functions
Body forces	Enabled

In the FLUENT computer program that the governing equations were discretized using the finite volume technique. The discretized equations, along with the initial and boundary conditions, were solved to obtain a numerical solution.

The model used for simulating the gas-solid flow is the Eulerian Multiphase Mixture Model (EMMM). The EMMM solves the continuity equation for the mixture, the momentum equation for the mixture, and the volume fraction equation for the secondary phase, as well as an algebraic expression for the relative velocity.

By using the mixture theory approach, the volume of phase q, V_q is defined by

$$V_q = \int_V \alpha_q \, dV \tag{1}$$

and $$\sum_{q=1}^{n} \alpha_q = 1 \tag{2}$$

The effective density of phase q is

$$\hat{\rho} = \alpha_q \rho_q \tag{3}$$

Where ρ_q is the physical density of phase

3.1 Conservative Equations

The general conservation equations from which the solution is obtained by FLUENT are being presented below:

3.2 Conservation of mass

The continuity equation for phase q is

$$\frac{\partial}{\partial t}(\alpha_q \rho_q)+\nabla.(\alpha_q \rho_q \vec{v}_q)=\sum_{p=1}^{n}\dot{m}_{pq} \qquad (4)$$

where $\vec{v}_q$ is the velocity of phase q and $\dot{m}_{pq}$ characterizes the mass transfer from the p[th] to q[th] phase.

From the mass conservation we can get:

$$\dot{m}_{pq} = -\ \dot{m}_{qp} \qquad (5)$$

$$\dot{m}_{pp} = 0 \qquad (6)$$

Usually, the source term ($\sum_{p=1}^{n}\dot{m}_{pq}$) on the right hand side of equation is zero.

3.4 Conservation of Momentum

The momentum balance for phase q yields

$$\frac{\partial}{\partial t}(\alpha_q \rho_q \vec{v}_q)+\nabla.(\alpha_q \rho_q \vec{v}_q \vec{v}_q)=-\alpha_q \nabla p+$$

$$\nabla.\bar{\tau}_q+\alpha_q \rho_q \vec{g}+\sum_{p=1}^{n}\left(\vec{R}_{pq}+\dot{m}_{pq}\vec{v}_q\right)+$$

$$\alpha_q \rho_q (\vec{F}_q+\vec{F}_{lift,q}+\vec{F}_{vm,q}) \qquad (7)$$

Where $\bar{\tau}_q$ is the q[th] phase stress-strain tensor.

$$\bar{\tau}_q=\alpha_q \mu_q (\nabla \vec{v}_q+\nabla \vec{v}_q^{\,T})+\alpha_q (\lambda_q-\frac{2}{3}\mu_q)\nabla.\vec{v}_q \bar{I} \qquad (8)$$

Here μ_q and λ_q are the shear and bulk viscosity of phase q, $\vec{F}_q$ is an external body force. $\vec{F}_{lift,q}$ is a lift force, $\vec{F}_{vm,q}$ is a virtual mass force, $\vec{R}_{pq}$ is an interaction force between phases, and p is the pressure shared by all phases.

$\vec{v}_q$ is the interphase velocity and I can be defined as follows.

If $\dot{m}_{pq} > 0$ (i.e., phase p mass is being transferred to phase q), $\vec{v}_{pq}=\vec{v}_p$; (9)

If $\dot{m}_{pq} < 0$ (i.e., phase q mass is being transferred to phase p),

$$\vec{v}_{pq}=\vec{v}_q; \qquad (10)$$

$$\vec{v}_{pq}=\vec{v}_{qp} \qquad (11)$$

The above equation must be closed with appropriate expressions for the interphase force $\vec{R}_{pq}$. This force depends on the friction, pressure, cohesion, and other effects, and is subject to the conditions that

$$\vec{R}_{pq}=-\vec{R}_{qp} \quad \text{and} \quad \vec{R}_{qq}=0 \qquad (12)$$

FLUENT uses the following form:

$$\sum_{p=1}^{n}\vec{R}_{pq}=\sum_{p=1}^{n}K_{pq}(\vec{v}_p-\vec{v}_q) \qquad (13)$$

Where $K_{pq}=K_{qp}$ is the interphase momentum exchange coefficient.

4. TURBULENCE MODEL

In order to account for the effects of turbulent fluctuations of velocities the number of terms to be modeled in the momentum equations in multiphase is large and this makes the modeling of turbulence in multiphase simulations extremely complex. The turbulence model used for the current simulations is based on Mixture Turbulence Model (MTM). The κ and ε equations describing ths model are as follows:

$$\frac{\partial}{\partial t}(\rho_m \kappa)+\nabla.(\rho_m \vec{v}_m \kappa)=\nabla.\left(\frac{\mu_{t,m}}{\sigma_\kappa}\nabla\kappa\right)+G_{\kappa,m}-\rho_m\varepsilon \qquad (14)$$

$$\frac{\partial}{\partial t}(\rho_m \varepsilon)+\nabla.(\rho_m \vec{v}_m \varepsilon)=\nabla.\left(\frac{\mu_{t,m}}{\sigma_\varepsilon}\nabla\varepsilon\right)+\frac{\varepsilon}{\kappa}(C_{1\varepsilon}G_{\kappa,m}-C_{2\varepsilon}\rho_m\varepsilon) \qquad (15)$$

Where the mixture density and velocity, ρ_m and $\vec{v}_m$, are computed from:

$$\rho_m=\sum_{i=1}^{N}\alpha_i\rho_i \qquad (16)$$

$$\vec{v}_m=\frac{\sum_{i=1}^{N}\alpha_i\rho_i\vec{v}_i}{\sum_{i=1}^{N}\alpha_i\rho_i} \qquad (17)$$

The turbulent viscosity, $\mu_{t,m}$, is computed from:

$$\mu_{t,m}=\rho_m C_\mu \frac{\kappa^2}{\varepsilon} \qquad (18)$$

and the production of turbulence kinetic energy, $G_{k,m}$, is computed from

$$G_{\kappa,m}=\mu_{t,m}(\nabla\vec{v}_m+(\nabla\vec{v}_m)^T):\nabla\vec{v}_m \qquad (19)$$

5. BOUNDARY CONDITIONS

At the inlet, all velocities and volume fractions of both phases are specified. The pressure is not specified at the inlet because of the incompressible gas phase assumption (relatively low pressure drop system). The initial velocity of gas and solid phase is being specified as mentioned in Table 1.

The meshing was done using Gambit 1.2. Fine meshing was done for riser inlet and exit sections in order to analyze them in a better way. Under relaxation factors were tuned to achieve convergence. The convergence tolerance was set at 0.001.

The main parameters of the flow inside the system are calculated using an iteration calculation procedure performed by FLUENT. An iterative cycle starts with the introduction of the initial data and/or initial guessed values, boundary conditions, physical conditions and constants. In a second step the program calculate the velocity field from the momentum equation. Then, the mass balance equations as well as the pressure equation are solved.

The next step is to update again the values of the parameters for both phases. The final step is to check on convergence which criterion is fixed by the user. If the convergence criterion is achieved the simulation will stop and give the final results of the system. If not, certain correction values are used to adjust the calculated values and the calculation will start all over again, using as initial data these last corrected values of each parameter.

The coefficient of restitution quantifies the elasticity of particle collisions. It has a value of 1 for fully elastic collisions and 0 for fully inelastic collisions. It is utilized to account for the loss of energy due to collision of particles, which is not considered in the classical kinetic theory. The restitution coefficient is close to unity. In this study, a particle-particle restitution coefficient of 0.95, and a particle-wall restitution coefficient of 0.9 were used [6].

6. RESULTS AND DISSCUSSIONS

A gross behaviour of a CFB is being presented here. Inward/outward motion, secondary flows of the first kind, tangential acceleration/deceleration, and cavity formation near riser exits is mechanisms can account for asymmetric flow in the exit region.

If more solids accumulate near the riser exit then fewer solids reside in the return leg. The lower rate of solids circulation may cause the solids volume fraction in the riser and connector to be lower. However, if the solids accumulation near the riser exit extends into these components, the solids volume fraction may be larger.

Fig. 2 indicates that a right angle exit with internal baffle and a blind T riser exits show that the solids volume fraction is more or less constant in the lower half of the riser. In the upper half, a strong increase of solids volume fraction with elevation was observed for the blind T exit, whereas a decrease is found for the right angle exit with the internal baffle. The size and shape of the upstream exit region is strongly dependent on the design of the riser exit.

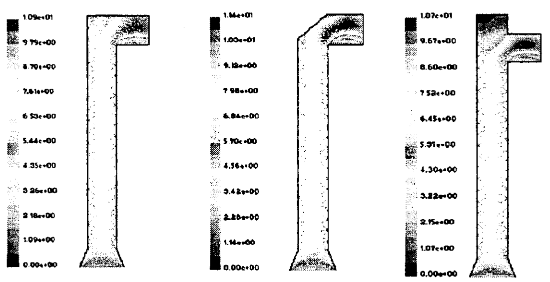

Fig.2 Contours of velocity profile in exit geometries

From Fig. 3, it can be inferred that the turbulence is more pronounced in the right angled and baffled exits. The blind T exit accumulated more solids than the right angle exit, and yielded a higher solids volume fraction in the riser. The solids hold-up is greater for the exit with baffle. The blind T exits shows larger solids volume fractions along the entire riser height, and an increase of solids volume fraction with elevation in the upper half of the riser.

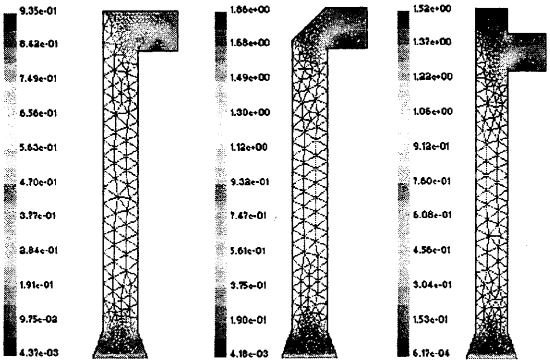

Fig 3. Contours of slip velocity

The solids volume fraction remains constant near the exit with internal baffle, but show an increase with elevation in the upper half of the riser for the right angle exit and blind T exit. The slip distribution in the various exits are different with right angle exit and with baffle showing greater slip than blind T exit.

All the above investigations suggest that riser exits can reduce solids hold-up in the riser and yield a region upstream where the solids volume fraction decreases with elevation. Riser exits yield an apparently unaffected solids volume fraction profile or increased solids hold-up and invoke a region where the solids volume fraction increases with elevation.

7. CONCLUSIONS

All the above investigations suggest turbulence in the riser as well its exits can reduce solids hold-up in the riser. Riser exits yield an apparently unaffected solids volume fraction profile or increased solids hold-up and invoke a region where the solids volume fraction increases with elevation. This suggests an upstream exit region to be defined as the region upstream of the riser exit where flow properties are affected by the riser exit. Similarly, a downstream exit region can be defined as the region downstream of the riser exit where flow properties are affected by the riser exit. The (overall) exit region is the region of the CFB where flow properties are affected by the riser exit, and comprises the upstream exit region, downstream exit region and the riser exit itself. The upstream exit region is generally characterized by a Core/Annulus structure, but that the solids mass flux profile may be asymmetric. Some riser exits appear to invoke regions near the riser wall where solids motion is upwards.

ACKNOWLEDGMENTS

We greatly acknowledge the financial support from UTM through Vote 74033 and Public Service Department, Malaysia. Assistance from Mr. Parag Vichare is also acknowledged.

REFERENCES

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

[2] Cheng Y., Wei F., Yang G. and Jin Y. (1998) *Powder Technology,* **98**, pp. 151-156.

[3] Gilbertson, M.A. and Yates, J.G. (1996) The Motion of Particles Near a Bubble in a Gas-Fluidized Bed, *Journal of Fluid Mechanics,* **323**, pp.377-385.

[4] Pain, C.C., Mansoorzadeh, S. and de Oliveira, C.R.E. (2001) A Study of Bubbling and Slugging Fluidised Beds Using the Two-Fluid Granular Temperature Model, *International Journal of Multiphase Flow*, **27**, pp. 527-551.

[5] Jalil R., Tasirin S. & M. S. Takriff (2002) Computational Fluid Dynamics (CFD) in Fluidized Bed column: effect of Internal Baffles, *The proceedings of RSCE Oct 2002,* Malaysia.

[6] Hansen, K. G., Madsen, J., Ibsen, C. H., Solberg T. & Hjertager B. H. (2002) An experimental and computational study of a gas-particle flow in a scaled circulating fluidized bed, *World Congress on Particle Technology*, Sydney, NW, Australia.