# Hydrodynamic Study of Fluidized Bed using Multiphase CFD

**Jaikishan Chandarana**

Email id: jai28.kc@gmail.com

**Abstract**

Internally Circulating Fluidized Bed (ICFB) is a type of Fluidized Bed Column where a draft tube is present for facilitating re- circulation of the packing material in the column. This ensures that any packing material is not lost at the outlet and is retained by the column which in turn decreases the replacement costs for the packing material. Heat and energy losses caused due to the packing material ejection is also eliminated. The system has a wide variety of applications in physical, chemical, petrochemical, electrochemical and biochemical processing. It can be used for purifying any fluid streams before exiting the system. Using ICFB, up to 95% of pollutants will be absorbed before being emitted to the atmosphere. At the same time, the advantages of this technology include capturing pollutants, perfect mixing, using alloys and plug free water supply, reduce maintenance costs, and low fuel consumption.

Hydrodynamic study of gas and solid flow in an ICFB is performed using the CFD multi-phase model. 2D computational meshes were used to represent physical ICFB geometries of 0.1 m and 0.0053 m diameter columns. The model approach uses the two-fluid Eulerian model with kinetic theory of granular flow options to account particle-particle and particle-wall interactions. The model also uses various drag laws to account the gas-solid phase interactions. The Gidaspow and Schiller - Naumann drag models predict the fluidization dynamics in terms of flow patterns, void fractions and axial velocity fields.

The experimental setup has a fluidized bed of height 1.88m and diameter 10cm. The gas (air) and liquid (water) are injected at the base with different velocities while taking glass beads of different diameters as solid bed. The variables investigated are pressure drop, gas holdup and bed expansion. The results of simulation are verified by comparing it with few experimental results and then rest of the prediction was done instead of carrying out the experiments. This helped in saving the experimental costs and prevented from risk of wastage of resources.

The work is divided mainly into four parts. The first part discusses about importance of gas-liquid-solid fluidized bed, their modes of operation, important hydrodynamic properties that have been studied either related to modelling or experimental analysis and applications of gas-liquid-solid fluidized bed. The second part gives an overview of the methodology used in CFD to solve problems relating to mass, momentum and heat transfer. Also, comparative study of various CFD related software is discussed. Third part gives the details about problem description and approach used in FLUENT to get the solution. Finally results of simulation and comparison with experimental results are shown.

*Keywords: ICFB; CFD Multiphase model; Perfect Mixing; Eulerian Model; Gidaspow and Schiller – Naumann; Hydrodynamic Properties; FLUENT;*

## 1. INTRODUCTION

In a typical gas–liquid–solid three-phase fluidized bed, solid particles are fluidized primarily by upward concurrent flow of liquid and gas, with liquid as the continuous phase and gas as dispersed bubbles if the superficial gas velocity is low. Because of the good heat and mass transfer characteristics, three-phase fluidized beds or slurry bubble columns ($u_t < 0.05$ m/s) have gained considerable importance in their application in physical, chemical, petrochemical, electrochemical and biochemical processing. Gas–liquid–solid fluidized beds are used extensively in the refining, petrochemical, pharmaceutical, biotechnology, food and environmental industries. Some of these processes use solids whose densities are only slightly higher than the density of water.

Two important hydrodynamic transitions within gas– liquid–solid fluidized beds are the minimum liquid fluidization velocity, $U_{Lmf}$, and the transition velocity from the coalesced to dispersed bubble regime, $U_{cd}$. The minimum liquid fluidization velocity is the superficial liquid velocity at which the bed becomes fluidized for a given superficial gas velocity; above the minimum liquid fluidization velocity, there is good contact between the gas, liquid and solid phases which is essential for heat and mass transfer processes. In the coalesced bubble regime, bubble size varies as the bubbles continuously coalesce and split, while in the dispersed bubble regime, there is no coalescence and thus the bubble size is more uniform and generally smaller.

Intensive investigations have been performed on three-phase fluidization over the past few decades; however, there is still a lack of detailed physical understanding and predictive tools for proper design, scale-up and optimum operation of such reactors. The calculation of hydrodynamic parameters in these systems mainly relies on empirical correlations or semi- theoretical models such as the generalized wake model and the structured wake model.

Though these models are capable of successfully elucidating the phenomena occurring in the three-phase reactors, too many parameters in them have limited their practical applications. In recent years, the computational fluid dynamics (CFD) based on the fundamental conservation equations has become a viable technique for process simulation.

## 2. PRESENT WORK

In the studies done so far, there has not been much emphasis on gas holdup and pressure drop. Here, the focus is on understanding the complex hydrodynamics of three-phase fluidized bed and internally circulating fluidized bed (ICFB) containing coarser particles of size above 1mm. The CFD software package FLUENT 6.2.16 has been used to simulate a solid-liquid-gas fluidized bed with a special designed air sparger aimed at improving the gas-liquid mixing in the distributor section and sending the well mixed gas-liquid mixture to the fluidizing section. The fluidized bed to be simulated is of height 1.88m and diameter 0.1m. The gas (air) and liquid (water) has been injected at the base with different velocities while taking glass beads of diameter 2.18mm as solid bed. The variables to be investigated are bed expansion, gas holdup and pressure drop.

CFD is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the millions of calculations required to simulate the interaction of fluids and gases with the complex surfaces used in engineering.

Advantages of CFD can be given as:

- It provides the flexibility to change design parameters without the expense of hardware changes. It therefore costs less than laboratory or field experiments, allowing engineers to try more alternative designs than would be feasible otherwise.
- It has a faster turnaround time than experiments.
- It guides the engineer to the root of problems, and is therefore well suited for trouble-shooting.
- It provides comprehensive information about a flow field, especially in regions where measurements are either difficult or impossible to obtain.

## 3. CFD SIMULATION OF GAS-LIQUID-SOLID FLUIDIZED BED

In the present work, a Eulerian multi-fluid model is adopted where gas, liquid and solid phases are all treated as continua, inter-penetrating and interacting with each other everywhere in the computational domain. The pressure field is assumed to be shared by all the three phases, in proportion to their volume fraction. The motion of each phase is governed by respective mass and momentum conservation equations.

*3.1 Continuity Equation*

$$\frac{\partial}{\partial t}(\varepsilon_k \rho_k) + \nabla(\rho_k \varepsilon_k \vec{u}_k) = 0$$

where $\rho_k$ is the density and $\varepsilon_k$ is the volume fraction of phase k= gas, solid, liquid and the volume fraction of the three phases satisfy the following condition:

$\varepsilon_g + \varepsilon_l + \varepsilon_s = 1$

*3.2 Momentum Equations*

*For liquid phase:*

$$\frac{\partial}{\partial t}(\rho_l \varepsilon_l \vec{u}_l) + \nabla(\rho_l \varepsilon_l \vec{u}_l \vec{u}_l) = -\varepsilon_l \nabla P + \nabla(\varepsilon_l \mu_{\mathrm{eff},l}(\nabla \vec{u}_l + (\nabla \vec{u}_l)^{\mathrm{T}})) + \rho_l \varepsilon_l g + M_{i,l}$$

*For gas phase:*

$$\frac{\partial}{\partial t}(\rho_g \varepsilon_g \vec{u}_g) + \nabla(\rho_g \varepsilon_g \vec{u}_g \vec{u}_g) = -\varepsilon_g \nabla P + \nabla(\varepsilon_g \mu_{\mathrm{eff},g}(\nabla \vec{u}_g + (\nabla \vec{u}_g)^{\mathrm{T}})) + \rho_g \varepsilon_g g - M_{i,g}$$

*For solid phase:*

$$\frac{\partial}{\partial t}(\rho_s \varepsilon_s \vec{u}_s) + \nabla(\rho_s \varepsilon_s \vec{u}_s \vec{u}_s)$$

where P is the pressure and $\mu_{eff}$ is the effective viscosity. The second term on the RHS of solid phase momentum equation is the term that accounts for additional solid pressure due to solid collisions. The terms $M_{il}$, $M_{ig}$, and $M_{is}$ of the above momentum equations represent the interphase force term for liquid, gas and solid phase, respectively.

*3.3 Problem Description*

The problem consists of a three-phase fluidized bed and Internally Circulating Fluidized Bed in which air and liquid (water) enters at the bottom of the domain. The bed consists of solid material (Glass beads) of uniform diameter which forms a desired height in the bed.

Table 1: Properties of phases used in experiment

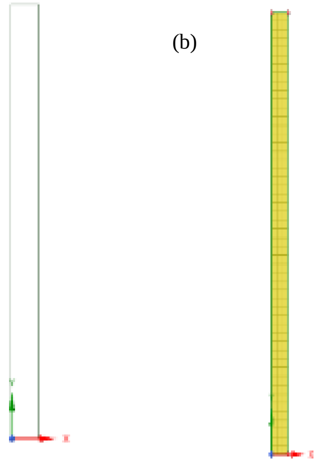
| Phases | Density (Kg/m$^3$) | Viscosity (Kg/m-s) |
|---|---|---|
| Gas – Air | 1.225 | 1.789 x $10^{-5}$ |
| Liquid – Water | 998.2 | 1.003 x $10^{-3}$ |
| Solid – Glass Beads | 2470 | Same as water |

*3.4 Simulation*

*3.4.1 Geometry and Meshing*

2D rectangular geometry with width of 0.1m and height 1.88m. Coarse mesh size of 0.01m was taken in order to have 1880 cells (3958 faces) for the whole geometry. Similarly, a mesh size of 0.005 cm was also used in order to have better accuracy. But using fine mesh results in 7520 cells (15436 faces), which requires smaller time steps, a greater number of iterations per time step and 4 times more calculation per iteration for the solution to converge. Also, because results obtained in case of coarse grid were in good accordance with experimental outputs, coarse grid was preferred over finer grid for simulation. In case of ICFB 2D rectangular outer geometry with width of 0.1m and height 1.88m. Coarse mesh size of 0.01m was made and internal bed of 2D rectangular geometry with width of 0.0053m and height 0.247m. Coarse mesh size of 0.01m.

(a) (b)



Figures 1(a) and (b): Geometry and Mesh of Fluidized Bed

Figures 2(a), (b) and (c): Geometry and Mesh of ICFB

*3.4.2 Selection of Models for Simulation*

FLUENT 6.2.16 was used for simulation. 2D segregated 1st order implicit unsteady solver is used. (The segregated solver must be used for multiphase calculations). Standard k-ε dispersed eulerian multiphase model with standard wall functions were used.

The model constants are tabulated as:

Table 2: Model Constants used in the experiment

| Constant | Value |
|---|---|
| $C_{mu}$ | 0.09 |
| C1-Epsilon | 1.44 |
| C2-Epsilon | 1.92 |
| C3-Epsilon | 1.3 |
| TKE Prandtl Number | 1 |
| TDR Prandtl Number | 1.3 |
| Dispersion Prandtl Number | 0.75 |

Water is taken as continuous phase while glass and air as dispersed phase. Inter-phase interactions formulations used were
Liquid – Air: schiller-Naumann
Solid-Liquid: Gidaspow
Solid-Air: Gidaspow
Air velocities ranging from 0.0125 m/s to 0.1 m/s with increment of 0.0125 and water velocities from 0 to 0.17 m/s with inlet air volume fractions obtained as fraction of air entering in a mixture of gas and liquid were the parameters used for boundary conditions.
Pressure outlet boundary conditions:
Mixture Gauge Pressure- 0 Pa.

*3.4.3 Solid and Liquid Boundary Conditions*

Backflow Total Temperature- 293. Backflow
Granular Temperature -0.0001. Backflow
Volume Fraction- 0.
Specified shear was set as X=0 and Y=0 for gas and solid whereas no slip condition for water was used.

*3.4.4 Solution*

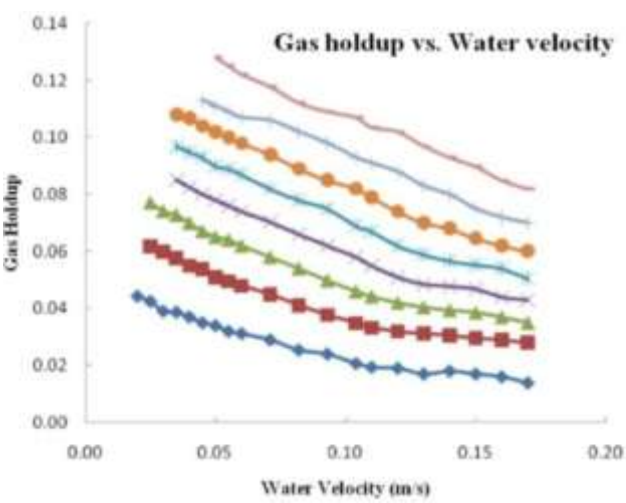
Under relaxation factor for pressure, momentum and volume fraction were taken as 0.3, 0.2, and 0.5 respectively. The discretization scheme for momentum, volume fraction, turbulence kinetic energy and turbulence dissipation rate were all first order upwind. Pressure-velocity coupling scheme was Phase Coupled SIMPLE. The solution was initialized from all zones. For patching a solid volume fraction calculated from experimental settings i.e., the volume fraction of solid in the part of the column up to which the glass beads were fed was used. Iterations were carried out for time step size of 0.01-
0.001 depending on ease of convergence and time required to get the result for fluidization.RESULTS AND DISCUSSION

A gas-liquid-solid fluidized bed of diameter 0.1m and height 1.88m has been simulated using commercial CFD software package FLUENT 6.2.16. The simulation has been done for static bed heights of 17.1cm and 21.3cm with glass beads of diameter 2.18mm. The inlet superficial velocity of air ranges from 0.0125m/s to 0.1m/s while that of water ranges from 0.0m/s to 0.14 m/s. The results obtained have been presented graphically in this section.

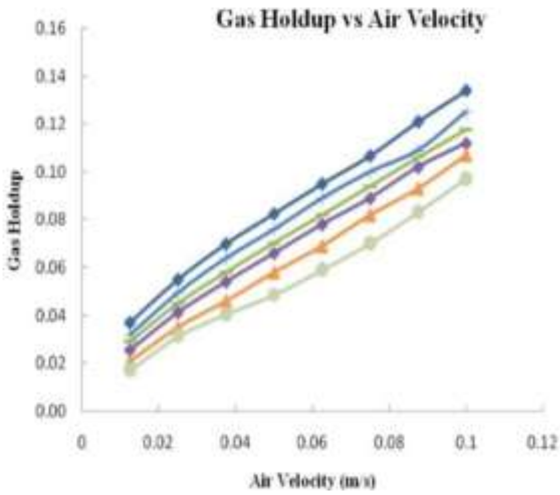
While simulating the fluidized bed the profile of bed changes with time. But after some time, no significant change in the profile is observed. This indicates that the fluidized bed has come to a quasi-steady state. Contours of volume fraction of bed with respect to time of fluidization are shown in fig 4. Simulation should be carried out till there no significant change in the profile of bed; like that between 20 and 30 seconds.

Gas holdup is the volume fraction of gas in the total volume of gas–liquid phase in the bubble column, which is one of the most important parameters to characterize the hydrodynamic characteristics of the bubble column.

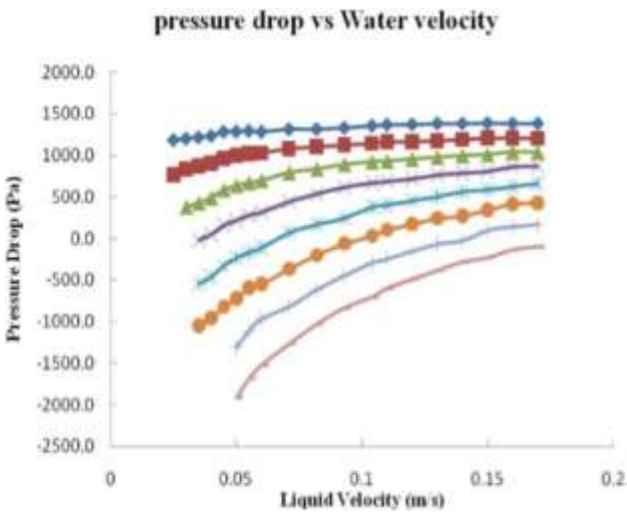
Graph 1: Gas holdup vs. Water Velocity



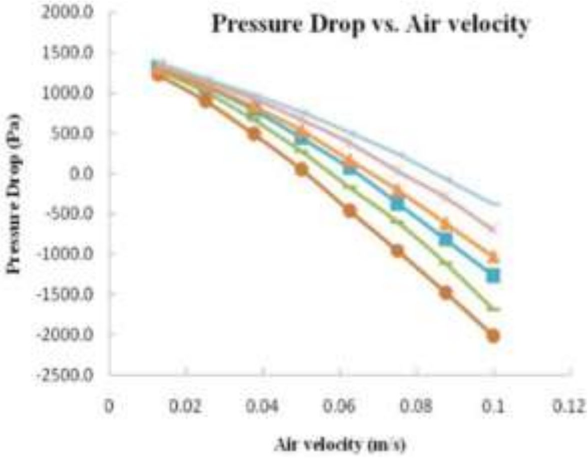

Graph 2: Gas holdup vs. Inlet Air Velocity



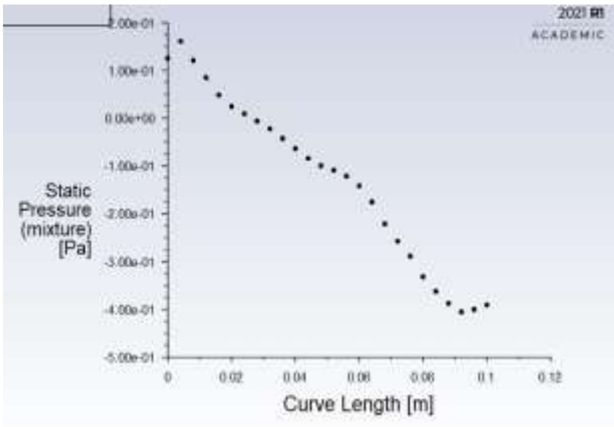

Graph 3: Pressure Drop vs. Water Velocity



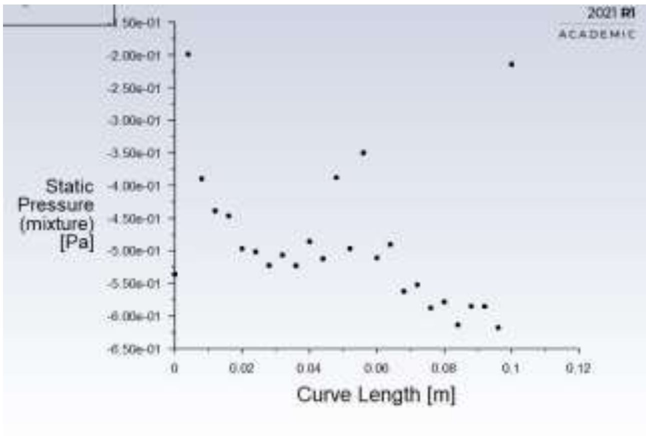

Graph 4: Pressure Drop vs. Air Velocity



Graph 5: Pressure variation in Fluidized Bed




Graph 6: Pressure variation in ICFB




As seen in the above graphs there are certain locations where the pressure is negative. This negative pressure is created below the rising bubbles. A narrow opening at the bottom allows the outside fluid to flow in the bed containing solids which produces vigorously fluidized bed. The small bubbles that are formed in the bed coalesce to become larger bed where pressure with a positive pressure rise, a negative pressure is created below them.

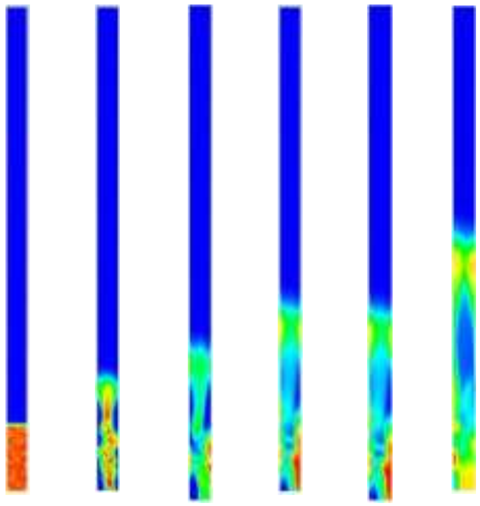


Figure 3: Fluidization occurring in a Fluidized Bed Column

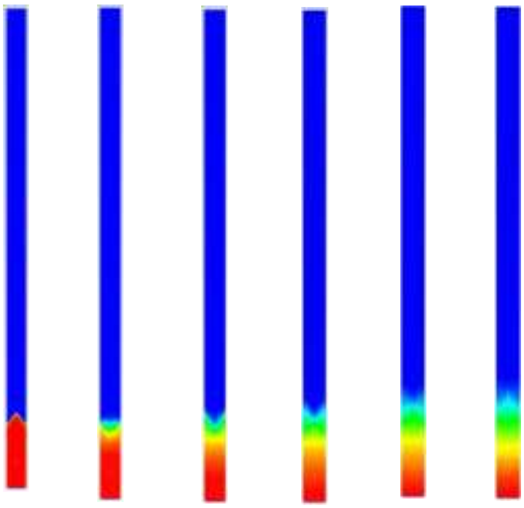


Figure 4: Fluidization occurring in an ICFB column

Results Obtained from the study and experimentation are as shown below:

- The figures 3 and 4 depicting the fluidization process show that the bed expands when the velocities of the fluid phases (air and water) increases.
- Trends of gas holdup vs. inlet water velocity obtained at different inlet air velocities show that gas holdup decreases when water velocity is increased. As liquid velocity increases volume fraction of gas reduces and hence the gas holdup decreases with increasing liquid velocity. Refer to Graph 1 for further analysis.

- Trends of gas holdup vs. inlet air velocity obtained at different inlet water velocities show that gas holdup increases when air velocity is increased. As air velocity increases volume fraction of air increases thereby gas holdup increases. Refer to Graph 2 for further analysis.

- Trends of pressure drop vs. water velocity obtained at different inlet air velocities show that pressure drop increases when water velocity is increased. Also, when the air velocity is small ($U_g$=0.0125 m/s) there is no substantial increase in pressure drop. This can be attributed to the fact that at small air velocity, the operation becomes almost solid-liquid fluidization because of small inlet volume fraction of the air compared to that of water. Refer to Graph 3 for further analysis.

- Plots of pressure drop vs. inlet air velocity obtained at different inlet water velocity show that pressure drop decreases as air velocity is increased. This is because with increase in air velocity the hold up of water in the column decreases (whose density is much larger compared to that of air) thereby decreasing the pressure drop across the column. Refer to Graph 4 for further analysis.

- The pressure is influenced by the initial static bed height. This is low for small initial bed height compared to higher initial bed height. Refer to Graph 5 and Graph 6 for further analysis.

- Since the bed expansion, gas holdup and pressure drop data obtained from simulation are showing very small deviation from experimental results, therefore CFD model developed in this work can be taken as very good approximation of gas-liquid-solid fluidized bed.

- In an internally circulating fluidized bed, and a traditional fluidized bed, the solids circulation rate from a fluidized bed to a moving bed were determined, along with the effects of gas velocity on solids circulation rate in the fluidized bed.

- The solids circulation rate increases with increase in gas velocity in ICFB. The simulation results demonstrate that the present CFD model can capture the key features of an ICFB system. Refer to Figure 4 for further analysis.

## 4. COMPARISON

*5.1 Fluidized Bed System*

Fluidized Bed System (FBS) is a type of device that can be used to carry out a variety of multiphase processes. In this type of device, a fluid (gas or liquid) is passed through a granular solid material (usually a catalyst possibly shaped as tiny spheres) at high enough velocities to suspend the solid and cause it to behave as though it were a fluid.

Table 3: Fluidized Bed Column' s Properties

| Advantages | Disadvantages |
|---|---|
| Uniform Particle Mixing | Increased Reactor Vessel Size |
| Uniform Temperature Gradients | Pumping Requirements and Pressure Drop |
| Ability to Operate Reactor in Continuous State | Particle Entrainment<br>Erosion of Internal Components<br>Pressure Loss Scenarios |

*5.2 Internal Circuit Fluidized Bed System*

The Internal Circulating Fluidized Bed (CFB) is a clean process with the ability to achieve lower emission of pollutants. By using this technology, up to 95% of pollutants will be absorbed before being emitted to the atmosphere. At the same time, the advantages of this technology outweigh the limitations, which made it favorable. With further R&D, this technology is predicted to bring numerous benefits to our society.

## 5. APPLICATIONS

Table 4: ICFB Column' s Properties

| Advantages | Disadvantages |
|---|---|
| Capture various pollutants.<br>Perfect Mixing.<br>Can use Alloys and Plug free water supply reduces maintenance costs.<br>Low fuel consumption. | Expensive manufacturing and fabrication.<br>High maintenance compared to fluidized beds. |

Gasification.
Combustion.
Fluid catalytic cracking. Production of melamine. Production of iron for steelmaking.
Fluidized roasting process of sulfide ore. TiO2 pigment production by chloride process. Fluidized bed drying.
Flue gas cleaning. Absorption and Desorption.
Extraction and Leaching processes.

## 6. CONCLUSION AND FUTURE SCOPE

The hydrodynamic characteristic of 2D FB and ICFB reactors gas-solid flow was studied by a Eulerian-Eulerian CFD model with the stress closer from kinetic theory of granular flow. Four different drag models were considered for the simulations. In case of ICFB the draft tube superficial gas velocity and the solids circulation rate have significant effect distribution of the solid volume fraction in each region. Increasing the draft tube superficial gas velocity can decreases solids volume fraction in the draft tube but has little effect in the annulus zone and decreases with an increase of solids concentration in the dense region ($\varepsilon > 0.1$).

This study has led to several conclusions on the study of CFB reactors. First of all, we have observed that a complete hydrodynamic simulation of the studied model is possible and relatively simple to carryout. The results we obtained are comparable to experimental data, and this can serve toward forwarding the development of this technology in the near future. Nevertheless, further study topics can be carried out for a more complete understanding of the model:

- Implement UDFs to further improve the model. It would be advantageous to finely control the introduction of char back in the riser through the loop seals with precise functions, as opposed to a simple constant reinjection rate, because this would allow us to more accurately simulate the separating and reinjection effect of the cyclone. This tends to show that the model is not yet completely realistic and sufficient in order to obtain the best simulation possible.

- Calculate heat extraction at the water lances. As observed in our results, the heat extraction aspect of the combustion simulation caused several issues which have to be resolved. In reality heat extraction occurs as a part of the normal functioning of the reactor.Further study all three loads of different hydrodynamic concepts, and of the effect of different inventories, fuel mass flow rates, gas velocities, etc., on the reactions.

## 7. REFERENCES

[1] CFD Modelling of Gas- Liquid-Solid Fluidized Bed - Amit Kumar in Chemical Engineering Department, NIT Rourkela, Odisha

[2] Internal Circulating Fluidized-bed Combustion Technology (ICFBC) - Japan Coal Energy Center; Ebara Corporation; Idemitsu Kosan Co., Ltd.

[3] Discrete phase simulation of gas–liquid–solid fluidization systems: single bubble rising behaviour - Jianping Zhang, Yong Li and Liang-Shih Fan in Powder Technology 113 2000 310–326

[4] A CFD study on the effect of bed thickness in an internally circulating fluidized bed - Quan Zhou, Junwu Wang, Yuking Feng and Peter J Witt in Conference Paper-January 2011

[5] Hydrodynamic Study of Gas-Solid Internally Circulating Fluidized Bed Using Multi-Phase CFD Model - Ravi Gujjala and Manadoddy Narasimha in Particulate Science and Technology-March 2015(Article)

[6] Eulerian–LaGrange simulations of liquid–gas–solid flows in three-phase slurry - Zhang and Ahmadi in Chemical Engineering Science 60 (2005) 5089– 5104

[7] The important role of local dispersed phase hold-ups for the calculation of three-phase bubble columns - Jörg Schallenberg, Jan H. Enß and Dietmar C. Hempel in Chemical Engineering Science 60 (2005) 6027– 6033