International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 12, pp: 90-95 www.ijaem.net ISSN: 2395-5252

# Experimental Investigation on Fluidized Bed Reactor Using CFD Model

BOOBESH MANI V[1]
*III Year Chemical Engineering, Adhiyamaan College Of Engineering*
*Ashok Kumar P[2], Chemical Engineering, Adhiyamaan College Of Engineering*
*Yuvaraj S[3], Assistant Professor, Adhiyamaan College Of Engineering*
*Corresponding Author[1]: boobeshmani2000@gmail.com*

Date of Submission: 14-11-2021 Date of Acceptance: 29-11-2021

**ABSTRACT:** Fluidized bed systems have the potential to be widely utilized in the facility generation, mineral dressing and chemical industries. One factor limiting their increased use is the lack of adequate design techniques for the scaling such systems. An experimental setup of fluidized bed has been fabricated with an internal diameter of 50mm with an air as fluidizing medium, with sand as a bed material. Bed pressure drop was measured as a function of superficial velocity. For the same setup, a CFD model has been developed for simulating gas-solid fluidized bed. The model uses a multi-phase Eulerian-Eulerian technique to predict the transient behavior of fluidized bed systems. The commercial CFD code FLUENT 6.3 is used as the computational frame work for solving the discredited equations. The model is used to predict hydrodynamics in a three-dimensional fluidized bed. The minimum fluidization velocity and pressure drop was compared with experimental values and found to be in aggregate with CFD results. It can be concluded that CFD is a valuable tool in fluidized bed design.

**KEYWORDS:** Fluidization, CFD, Fluent, VOF Model, Mixture Model, Eulerian Model

## I. INTRODUCTION

Fluidization is the popular means of contacting solids and a gas. Fluidization is that the operation by which fine solids are transformed into a fluid like state through contact with a gas or liquid.

Fluidization technology is found in many industries including petrochemical, power generation, drying, polymer production, environmental cleaning and scrubbing operations. Fluidized beds have many advantages including uniform temperature distribution, continuous operation, high mass transfer rates between gas/liquid and solids phases.

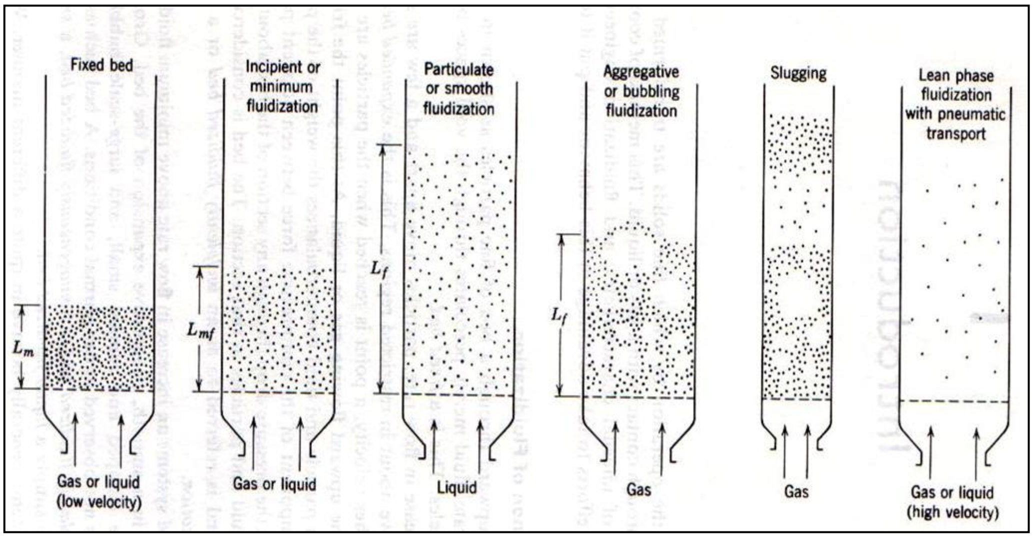



**Fig.1 Various Kinds of Contacting of Batch of Solids by Fluids is Shown Below**

**TYPES OF FLUIDIZATION**

**If the flow rate of the fluid is increased above the minimum required to produce a fluidized bed two types of fluidization will occur.**

(a) **Particulate Fluidization**

**In liquid-solid systems an increase in flow rate above minimum fluidization usually results in a smooth, progressive expansion of the bed. Gross flow instabilities are damped and remain small, and large-scale bubbling or heterogeneity is not observed under normal conditions. A bed such as this is called a *particulately fluidized bed, a homogeneously fluidized bed, a smoothly: fluidized bed*, or simply a *liquid fluidized bed*.**

(b) **Aggregative Fluidization**

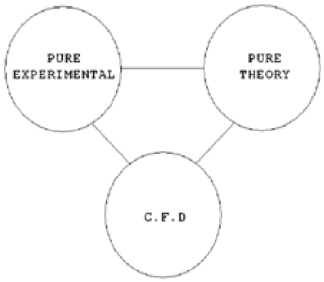
**Gas-solid systems generally behave in quite different manner. With an increase in flow rate beyond minimum fluidization, large instabilities with bubbling and channeling of gas are observed. At higher flow rates agitation becomes more violent and therefore the movement of solids becomes more vigorous. In addition, the bed doesn't expand much beyond its volume at minimum fluidization. Such a bed is called an aggregative fluidized bed, a heterogeneously fluidized bed, a bubbling fluidized bed, or simply a gas fluidized bed.**

Over the past half-century, we have witnessed the rise in the new methodology for attacking complex problem in fluid mechanics, heat transfer and combustion. It has come to the state that wherever there is a flow, computer can help to understand and analyze the same. This new methodology of solving a flow problem using a computer is given the name CFD. Computational Fluid Dynamics or CFD is that the analysis of systems involving fluid flow, heat transfer and associated phenomena like chemical reactions by means of computer-based numerical approach. Some important application areas are

- Aerodynamics of aircraft and vehicles: Lift and drag
- Power plant: Combustion In IC engines and gas turbines
- Fluidized beds : Combustion and gasification
- Turbo machinery: F1ows inside rotating passages, diffusers etc.
- Electrical and electronic engineering: cooling of equipment including micro- circuits
- Chemical process engineering: mixing and separation, polymer moulding
- External and internal environment of buildings: Wind loading and heating ventilation
- Marine engineering: Loads on off-shore structures, Hydrodynamics of ships
- Environmental engineering: Distribution of pollutants and effluents
- Hydrology and oceanography: Flows in rivers, estuaries, oceans
- Meteorology: Weather prediction
- Biomedical engineering: Blood flows through veins

Increasingly CFD is becoming vital component within the design of business products and processes.

**Analysation of a Fluid Flow Problem:**




There are three methods to research a fluid flow problem.

1. Experimental
2. Theoretical
3. Computational (CFD)

| *Approach* | *Advantages* | *Disadvantages* |
|---|---|---|
| Experimental | 1.Capable of being most realistic | 1. Equipment required<br>2. Scaling problems<br>3. Tunnel corrections<br>4.Measurement difficulties<br>5. Operating costs |
| Theoretical | 1.Clean,general,information, which is usually in formula form | 1. Restricted to simple geometry and physics<br>2. Usually restricted to linear problems |

| | | |
|---|---|---|
| Computational | 1.No restriction to linearity<br>2.Complicated physics can be treated<br>3.Time evolution of flow can be obtained | 1. Truncations errors<br>2. Boundary conditions problems<br>3. Computer costs |

**Comparison of CFD approaches**

**STEPS FOR SOLVING A PROBLEM USING FLUENT 6.3**

Various steps involved in solving a problem

- Reading the Grid
- Checking the Grid
- Displaying the Grid
- Selecting the Solver Formulation
- Defining Physical Models
- Specifying Fluid Properties
- Adjusting Solution Controls
- Specifying Boundary Conditions
- Saving the Case File
- Solving the Problem Initializing the Flow
- Calculating/Iterating
- Saving the Results
- Plotting the Contour

## II. EXPERIMENTAL SETUP

A schematic of experimental apparatus is depicted in below figure. A bed of chosen granular materials is contained in a vertical cylinder. At the lower end of the cylinder is a distribution chamber and an air distributor which supports the bed when defluidizied. This distributor has been designed to ensure uniform air flow into the bed without causing excess pressure drop. Air from the blower is delivered through a Rota meter fitted with a control valve, to distribution chamber One liquid filled manometer fitted to the apparatus. It displays the pressure of the air at any level in the bed chamber. An important feature of this unit is the ease with which the bed material may be changed. After unscrewing two union nuts to remove the air connections to the distribution chamber, three knurled nuts are removed from mounting bracket. The chamber, filter and distribution assembly may now be removed and also the bed material tipped out. Another bed material may now be poured into the cylinder. Once reassembling the components into reverse order, the nut can be operational with in two or three minutes.

The fluidized bed test rig is shown in below figure. The bed is of cylindrical shape of internal diameter 50 mm and maximum height 600 mm. The particles of sand residue are supported by a perforated plate. The fluidized gas (air) is supplied by a blower. The quantity is regulated by a gate valve. The air flow rate measured by an Rota meter. The pressure drop $\Delta H$ measured by U-tube manometers. All experiments are carried out at ambient pressure and temperature.

## III. EXPERIMENTATION

CFD codes are structured around the numerical algorithms. Nowadays all commercial CFD Packages include sophisticated user interfaces to input problem parameters and examine the results. All code contains three elements.

- A pre-processor
- A solver
- A post processor

Fluent is one of many CFD packages that are commercially available for the tackling of engineering problems involving Fluids. It can also handle problems involving Heat-Transfer, Combustion/Chemical Reactions Explosions etc.

Fluent uses finite difference numerical procedures to solve the governing equations for fluid velocities, mass flow, pressure, temperature, species concentration and turbulence parameters and fluid properties. Numerical techniques involve the sub-division of the domain into a finite set of neighboring cells known as "control volumes" and applying the discredited governing partial differential equations over each cell. This yields a large set of simultaneous algebraic equations which are highly non-linear. These equations are in turn solved by iterative means until a converged solution is achieved.

The criteria of convergence can be changed by the user, and is generally applied to the changes in the values of all the field variables from one iteration to the next. When all the equations are satisfied on all the discretisation points there'll be no change from one iteration to the subsequent. This theoretical convergence is not normally achievable in a finite number of steps. Hence the selection of suitable criteria to detect near convergence becomes important.

**STEPS FOR SOLVING A PROBLEM USING FLUENT 6.3**

Various steps involved in solving a problem

- Reading the Grid
- Displaying the Grid
- Selecting the Solver Formulation
- Defining Physical Models
- Specifying Fluid Properties
- Adjusting Solution Controls
- Specifying Boundary Conditions
- Checking the Grid
- Saving the Case File
- Solving the Problem Initializing the Flow
- Calculating/Iterating
- Saving the Results
- Plotting the Contours

**The Euler-Lagrange Approach:**

The Lagrangian discrete phases model in FLUENT follows the Euler-Lagrange approach. The fluid phase is treated as a continuum by solving the time averaged Navier-Stokes equations, while the dispersed phase is solved by tracking a large number of particles, bubbles, or droplets through the calculated flow field. The dispersed phase can exchange momentum, mass, and energy with the fluid phase. A fundamental assumption made in this model is that the dispersed second phase occupies a low volume fraction, even though high mass loading ($m_{particles} >= m_{fluid}$) is acceptable. The particle or droplet trajectories are computed individually at specified intervals during the fluid phase calculation. This makes the model appropriate for the modeling of spray dryers, coal and liquid fuel combustion, and some particle-laden flows, but inappropriate for the modeling of liquid-liquid mixtures, fluidized beds, or any application where the volume fraction of the second phase is not negligible.

**The VOF Model:**

The VOF model is a surface-tracking technique applied to a fixed Eulerian mesh. It is designed for two or more immiscible fluids where the position of the interface between the fluids is of interest. In the VOF model, a single set of momentum equations is shared by the fluids, and the volume fraction of each of the fluids in each computational cell is tracked throughout the domain. Applications of the VOF model include stratified flows, free-surface flows, filling, sloshing, the motion of huge bubbles during a liquid, the motion of liquid after a dam break, the prediction of jet breakup (surface tension), and therefore the steady or transient tracking of any liquid-gas interface.

**The Mixture Model**:

The mixture model is meant for 2 or more phases (fluid or particulate). As within the Eulerian model, the phases are treated as interpenetrating continua. The mixture model solves for the mixture momentum equation and prescribes relative velocities to explain the dispersed phases. Applications of the mixture model include particle-laden flows with low loading, bubbly flows, sedimentation, and cyclone separators. The mixture model also can be used without relative velocities for the dispersed phases to model homogeneous multiphase flow.

**The Eulerian Model:**

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 12, pp: 90-95 www.ijaem.net ISSN: 2395-5252

The Eulerian model is that the most complex of the multiphase models in FLUENT. It solves a group of n momentum and continuity equations for every phase. Coupling is achieved through the pressure and interphase exchange coefficients. The way during which this coupling is handled depends upon the sort of phases involved; granular (fluid-solid) flows are handled differently than non-granular (fluid-fluid) flows. For granular flows, the properties are obtained from application of kinetic theory of gases. Momentum exchange between the phases is additionally dependent upon the sort of mixture being modeled. FLUENT's user-defined functions allow you to customize the calculation of the momentum exchange. Applications of the Eulerian multiphase model includes bubble columns, risers, particle suspension, and fluidized beds.

## IV. OBESERVATIONS FROM THE EXPERIMENT

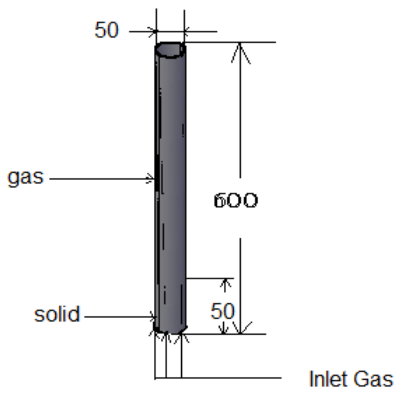
A transient three-dimensional, multi-phase fluidized bed has been developed using the commercial CFD code, FLUENT 6.3, as the computational engine. Close coupling between the two phases requires the use of small-time steps and consequently large CPU time requirements.

Results presented in this paper show that CFD technique is capable of predicting typical behavior such as minimum fluidization velocity, pressure drop and fluid motion observed in complex fluidized bed systems.

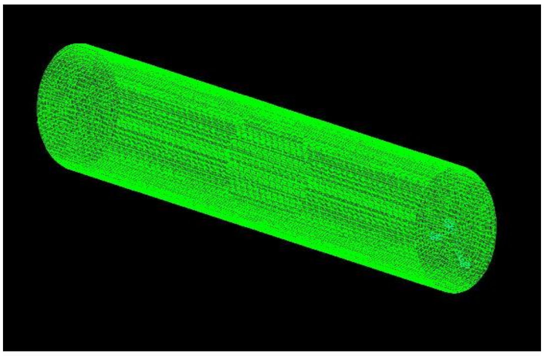
Recent experimental results are in reasonable qualitative agreement with predictions the model.

| S.no | name | Theoretical | Experimental | CFD |
|---|---|---|---|---|
| 1 | Minimum fluidization velocity($u_{mf}$)m/s | 0.132 | 0.15 | 0.13 |
| 2 | Pressure drop (pa) | 765.87 | 830 | 780 |

**FLUENT Contour Plots**

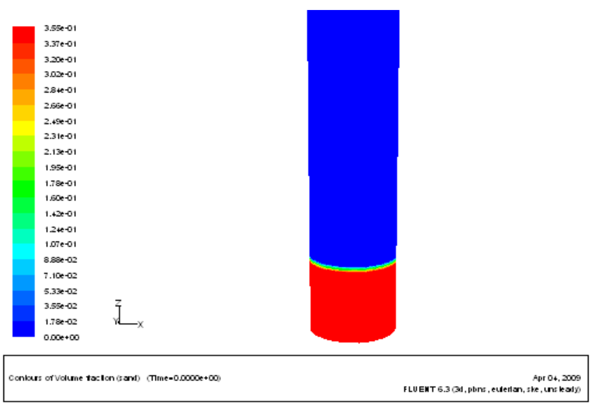



PROBLEM SPECIFICATION

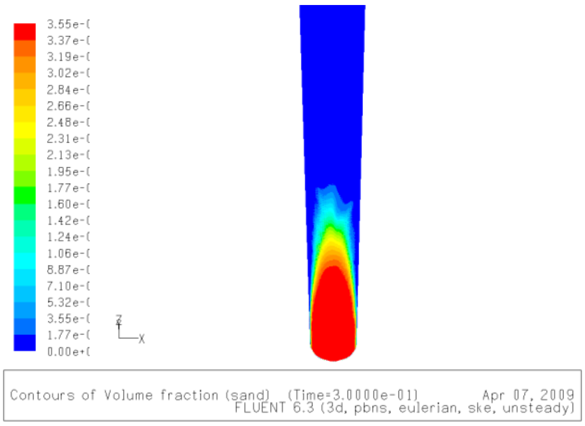


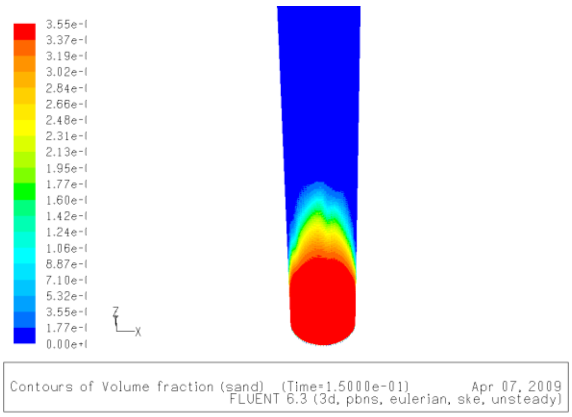
Grid of Problem

Contours of volume fraction (solid) at t=0sec



International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 12, pp: 90-95 www.ijaem.net ISSN: 2395-5252




Contours of volume fraction (solid) at t=0.15sec
Contours of volume fraction (solid) at t=0.3sec

## V. CONCLUSION

CFD technique is capable of predicting typical behavior such as minimum fluidization velocity, pressure drop and fluid motion observed in complex fluidized bed systems.

**SOME OF THE ADVANTAGES FROM THE ABOVE RESULTS**

a) Development cost reduction:
Using physical experiments and tests to get essential engineering data for design can be expensive
CFD simulations are relatively inexpensive, and costs are likely to decrease as computers become more powerful.
b) Quick assessment of design variations:
CFD simulations can be executed in a short period of time.
Engineering data can be introduced early in the design process.
c) Comprehensive information:
Experiments only permit data to be extracted at a limited number of locations in the system (where sensors and gauges are placed).
CFD allows the designer to examine any location in the region of interest, and interpret its performance through a set of thermal and flow parameters.
d) Enables the designer to simulate different conditions:
Many flow and heat transfer processes cannot be easily tested.
CFD provides the ability to theoretically simulate any physical condition.
CFD allows great control over the physical process, and provides the ability to isolate specific phenomena for study

## REFERENCES

[1]. Fluent 6.3- user's guide
[2]. Delgado J, Aznar MP, Corella J. Biomass gasification with steam in fluidized bed: effectiveness of CaO, MgO and CaO–MgO for hot raw gas cleaning. Ind. Eng. Chem. Res.1997;36:1535–43.
[3]. M'chirgui A, Tadrist L, Pantaloni J. Influence of particle-size distribution on entrainment solid rate in fluidized beds. Powder Technology 1993;77(2):177–99.
[4]. Howard JR. Fluidized bed technology—principles and applications. Bristol, New York: Adam Hilger1989. p. 1–69.
[5]. CheremisinoR NP, CheremisinoR PN. Hydrodynamics of gas–solids fluidization. Honston: Gulf Publishing Company, 1984. p. 2–10, 137–61