Case study to select an optimum CFD technique for a flow in a gas-solid fluidized bed

Dhananjay Ambre[1], Shubham Thakare[2], Abhishek Kumar[3] and Bhagwat Biradar[4]
Nilesh Birajdar[5]
†Mechanical Engineering, MITCOE Pune, Savitribai Phule Pune University, Pune, Maharashtra, India.

Abstract

This paper presents a case study for the selection of an optimum CFD technique from various techniques which are used to study and understand the flow in a Gas-Solid fluidized bed. Fluidization is an operation through which fine solid particles are transformed into a fluid like state through contact with either a gas, liquid or both and behave like a fluid through suspension in a liquid or gas. Fluidization is the preferred method of operation due to its many advantages over other configurations, like good solid mixing leading to uniform temperature and maintaining it throughout the bed, high mass and heat transfer rates, significantly lower pressure drops which reduce pumping costs and lower pollutant emissions, among many others. That is why, the Gas–Solid fluidized bed reactors have various advantages that make them attractive for a number of industrial processes. There is a need to improve the designs and reduce the emissions of these Gas-Solid fluidized bed reactors due to the increasing applications of the same. The fluidization and heat transfer behaviors of a fluidized bed are studied using various Computational Fluid Dynamics (CFD) techniques and flow models used for gas-fluidization.

Keywords: *Fluidization, Combustion, Multiphase flow, Eulerian-Eulerian, Eulerian-Lagrangian.*

1. Introduction

1.1 History

Fluidization gained more and more attention after it first appeared in commercial use (Geldart, 1973) in 1926. The new technique achieved widespread use soon in many industrial applications, especially in the fields of catalytic cracking and coal gasification, because of pressure during the wartime and a desperate need for aviation gasoline. Since then fluidization has always lived up to the expectations, turning into a well-established technology nowadays employed in many other areas such as, to cite just a few, coal combustion, biomass gasification and waste disposal. Meanwhile, with the advent of development in computer technologies, advanced commercial Computational Fluid Dynamics (CFD) software became available. CFD has become an essential tool in solving many complex and tedious problems of academic and industrial interests in vast areas for academics, researchers and engineers alike. These vast areas include fluidization, combustion, oil flow assurance, as well as aerospace science. CFD has enabled us to understand the basics of fluid-solid interaction. It has also helped us to correctly predict the different reactions seen in fluidized bed.

1.2 Fluidization

Fluidization is a process where a solid material is converted to a fluid-like state. This is done by allowing a fluid to flow through the solid material. This gives a condition where the solid particles suspend freely, when the gravitational forces on solid particles become equal to the fluid drag on them. This free suspension condition takes place at the critical value of fluid velocity. At this critical value, fluidization occurs and the bed is said to be fluidized which shows fluidic behavior.

In a fixed bed, when the fluid (gas or liquid) is passed upward, at a low flow rate, through a bed of fine particles, it merely percolates through the void spaces between the stationary particles. In the expanded bed, the flow rate is further increased such that particles move apart and a few are seen to vibrate and move in restricted regions. In the fluidized bed, where the fluid is at a higher velocity as compared to the fluid in expanded bed, a maximum point of pressure drop through the bed is reached where all the particles are suspended in the flowing fluid. First, the fluidization occurs at the bottom of the bed and after that it extends towards the top which results in a sharp decline in pressure drop.

When fluidization is initiated, three conditions are observed. First, the effective weight of the particle is neutralized by the force exerted between a particle and fluidizing medium. Second, adjacent particles lose their vertical component of the compressive force which existed in between them. Third, the weight of fluid and particles in a particular section becomes equal to the pressure drop through any section in the bed. These conditions in the bed are referred to as the bed at minimum fluidization. Under the assumption that friction is negligible between the particles and the bed walls, it is assumed that the vertical velocity of the fluid is uniformly distributed on the cross-sectional area and the lateral velocity of fluid is relatively small and can be neglected. The different stages of the principle of fluidization is shown in Fig 1.

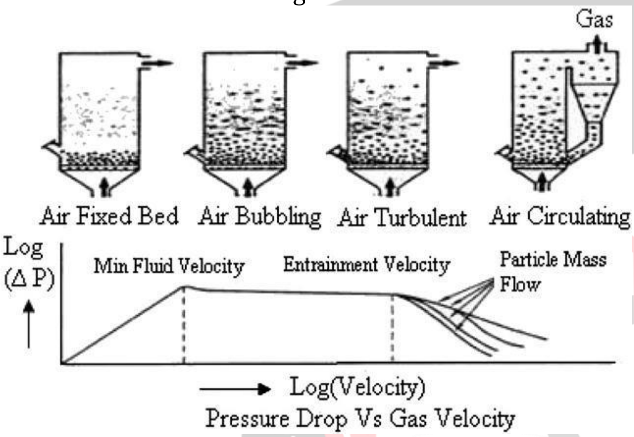

Fig 1- Principle of fluidization (Bureau of Energy Efficiency, 2005)

1.3 Fluidization techniques

Simulating and investigating directly the behavior of full-size systems without the need to draw on uncertain results obtained from experimentation, is a major advantage of CFD, for which it becomes an invaluable tool for research. But, for getting accurate results and making effective use of CFD, it is critical that accurate models must be developed along with appropriate constitutive equations.

Sound mathematical models which are able to predict the behavior of dense fluidized suspensions with acceptable accuracy have been developed with a lot of intensive fundamental research. There are three numerical techniques used for the studying combustion and gasification in fluidized beds in literature and these are Eulerian-Lagrangian, where the motion of each particle is tracked and the dynamics of the fluid is solved at a length scale much smaller than the mean particle diameter (Pan *et al*, 2002); Eulerian-Eulerian Two Fluid Model (TFM), which is based on the averaged equations of motion and treat the fluid and solid phases as two interpenetrating continua (Anderson and Jackson, 1967; Drew, 1971, 1983; Drew and Lahey, 1993; Drew and Segel, 1971; Enwald *et al*, 1996; Jackson, 1997, 1998, 2000; Whitaker, 1969; Zhang and Prosperetti, 1994); and Discrete Element Method (DEM-CFD) within Eulerian-Lagrangian concept.

1.4 Need for optimum fluidization technique

There is a need for effective use of the fossil fuels due to major concerns like fast depleting fossils fuels, energy security and environmental concerns. For this reason, fluidized bed combustion is considered as an important technology to control the greenhouse emissions, among many other clean coal technologies.

The major advantages of fluidization include low combustion temperature, low pollutant emissions, high efficiency, and high heat transfer rates which make fluidized bed combustors and gasifiers suitable and important for their use in many chemical and power industries. These applications of fluidized bed combustors and gasifiers are developing at fast pace in the power generating industry as they combine fuel flexibility and high efficiency especially for biomass co-combustion.

Combustion and gasification in fluidized bed can be studied by CFD, which is an economical and effective modelling tool. Predictions of temperature profiles of solid and gaseous phase present in dense bed, fuel mixing efficiency, inert material concentration in bed, temperature profile of furnace, heat flux, etc. are essential for the optimization of fluidized bed unit's design. These predictions are easily found out by reliable CFD models. The control of efficient operation requires prediction of such critical parameters which is done by simulation with aid of CFD. In the design process of large-scale fluidized bed, substitution of empirical or semi-empirical models with the aforementioned CFD techniques is expected.

Fluidized bed combustion and gasification is a multiphase reactive flow phenomenon. It involves homogeneous reactions among gases and heterogeneous reactions between fuel particles and gases, which concludes it as a multiphase problem between gases and fuel particles and it is also a reactive flow problem (Singh *et al*, 2013). The fuel in the fluidized bed for combustion and gasification reaction exists in all the three states form in the sand particles where the steam-air in the case of gasification and gasifying agent is air in combustion. The modelling of these fluidized bed devices is extremely complicated due to the involvement of gasification, combustion and multiphase flow, which continues to be a challenge to the practicing engineers and scientific community.

2. Literature Review

In earlier times, most of the literature of CFD studies on fluidized bed devices had a focus on isothermal modelling of dense bed (Gnanapragasam and Reddy, 2009, Chen *et al*, 2011, Wang *et al*, 2010, Gao *et al*, 2007, Behjat *et al*, 2008). Only due to the advancement in computational technology, it is possible to study the combustion and gasification in fluidized bed by CFD modelling. Researchers tried to use CFD to study fuel, char, ash, physical and chemical behavior in fluidized bed. They also used CFD in studying operational parameters, nitrogen chemistry, emissions, co-firing coal with biomass, ash deposition behavior prediction, conversion of fuels in bed and freeboard/riser, calcination, and non-commercial technologies like chemical looping combustion systems.

Fluidized bed combustor and gasifier devices could be divided mainly into three parts based on the geometric regions of fluidized bed furnace for the numerical modelling of the bed and devices. They are, namely, dense bed, splash zone and freeboard/riser of fluidized bed units. The Eulerian-Eulerian (E-E) (TFM) approach is used for the dense bed where most of the studies are concentrated on gasification and simple geometries. CFD for combustion of coal in circulating fluidized bed combustor overlooking three-dimensional effects was considered by few of them (W. Zhou *et al*, 2011; W. Zhou *et al*, 2011; W. Zhou *et al*, 2009). The three-dimensional or full-scale device geometry to investigate the unit with E-E approach considering combustion/gasification occurring in bed was considered by only two authors (Nikolopoulos *et al*, 2009; Myöhänen *et al*, 2006).

The region above the dense bed, i.e. freeboard region, where the diluted particle conditions are present, the other CFD technique, i.e. Eulerian-Lagrangian is used. Different authors tried to study freeboard in fluidized beds by touching different aspects, by applying CFD to study combustion and gasification issues of solid fuels, their emissions, operational parameters and other aspects like fate of nitrogen in freeboard (A. Brink *et al*, 2001; A. Brink *et al*, 2006; A. Brink *et al*, 2005; R. Zevenhoven *et al*, 2001). Some used CFD by using their own code and Fluent to find the temperature, flow and main species in the fluidized bed combustor. Some others used CFD in commercial fluidized bed combustors to try to find ash deposition prediction (Mueller *et al*, 2005; Mueller *et al*, 2001; Mueller *et al*, 2003; Mueller *et al*, 2002; Lundmark *et al*, 2007; Lundmark *et al*, 2010). They got boiler surfaces with deposition maps along with high probability of ash position. Some authors tried using probability density approach to model the freeboard (Ravelli *et al*, 2008; Rozainee *et al*, 2010, Agraniotis *et al*, 2009). They considered burning of the fuel in gas phase above the bed. By integrating the force balance on the fuel particles, written in Lagrangian form, we get the tracking of fuel particles in a discrete phase with DPM.

(Brink *et al*, 2009), considered the splashing zone of a bubbling bed where he analyzed the behavior of biomass particles by forming a simplified model. According to (Myöhänen *et al*, 2006), overlooking of three-dimensional behaviors is done in most of the literature. (Yuxin *et al*, 2004) concluded that in the various fields of fluidized bed like combustion, emissions performance and numerical prediction of hydrodynamic behavior, new challenges appeared.

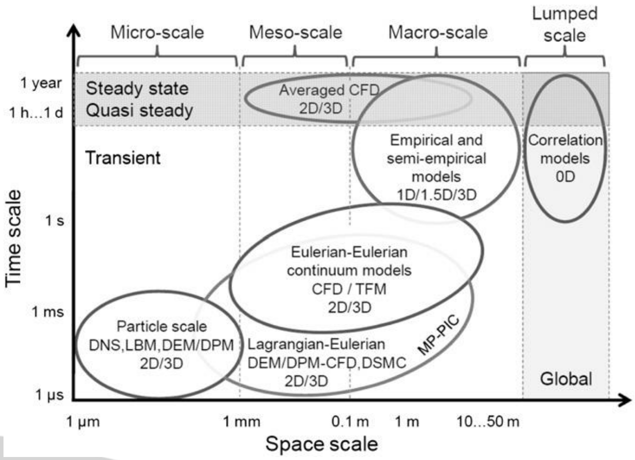

Fig 2-Scale-based classification of multiphase approaches for fluidized bed (Myohanen *et al*, 2011).

Quantitative predictions of temperature profiles or the gas concentration inside the bed are not possible by many studies. But, the CFD models, taking into account the combustion/gasification issues in fluidized bed, are capable of predicting qualitative as well as quantitative information. Most of the quantitative results belong to E-L DPM model which are used to study the temperature profile, heat transfer or emissions of freeboard/riser. But, the tracking of each particle with the Lagrangian method is not practical with the current computational capacity, so E-E TFM method is adopted in most of studies. Moreover, in the Lagrangian method, the interactions between particles and gases are limited to a single particle or diluted particle concentration conditions which can't be used for dense condition in a fluidized bed. The application of E-E TFM in commercial boilers considering thermo-chemical reactions and heat exchange coefficients between solids (i.e. inert material and fuel) in dense beds is not found in any paper.

2.1 Eulerian-Eulerian: -

(Yu *et al*, 2007) performed numerical simulations of the bubbling fluidized bed coal gasification for two-dimensional bubbling fluidized bed gasifier (BFBG) is done by. They developed their own code for a 2-D k-ε model without using any User Defined Functions (UDF). The simulation results they got would not have been possible to get from two-phase or three-phase one dimensional fluidization models. The results included accurate predictions of the temperature, distributions of pressure, volume fraction of the phases, velocity and gas composition.

(Jung and Gamwo, 2008) applied multiphase CFD based models for chemical looping combustion process using MFIX code and did fuel reactor modelling. UDF were used for the 2-D model where methane and NiO were used as catalyst in the fuel reactor. The solutions and simulations agreed with the experimental data.

(Deng *et al*, 2008) used multiphase CFD modelling for a chemical looping combustion process. In this work, the reaction kinetics models of the ($CaSO_4$ + H_2) fuel reactor are developed. They also studied effect of various parameters like flow rate, particle diameter, bed temperature, etc. on performance along with validation of model. They concluded that the conversion of H_2 increases at higher temperatures and decreases with particle size.

(Zeng *et al*, 2008) discussed a 3-D CFD model of coal gasification in fluidized bed and the effect of various parameters on gasification parameters, i.e. emissions. The shrinking core model was run on Fluent software with k- ε as the turbulence model. They got two results, first, the increase of bed temperature enhanced the formation of all other gases except H_2 and CH_4 and second, the gas quality (combustible fractions and caloric value) improves at a higher operating pressure. They explained that the gasification rate enhanced directly by pressure due to the increase in the partial pressure of reactants and other as the fluidization in the reactor becomes better at elevated pressure. Experimental verification was done on 0.1 MW reactor and the simulation results agreed with the experimental results.

The development and testing of a multiphase 2-D CFD model for chemical looping combustion is done by (Emden *et al*, 2008). The model is a linear and spherical shrinking core with k-ε turbulence model which was run on Ansys Fluent. The chemical looping was operated with methane as fuel gas and Mn_3O_4 as oxygen carrier. The computational results, which were achieved by applying a UDF, did not agree with the experimental simulations, as it could not apprehend the direct proportionality between the methane conversion and fuel injection rate.

A Eulerian CFD modelling approach of wood gasification in a bubbling fluidized bed reactor using char as bed material is done by (Gerber *et al*, 2010). They have formed the E-E TFM CFD model for char as bed material in two-dimensional fluid bed gasifier and used an open-sourced code for simulation. They studied the various parameters like effect of initial bed height, variations in fuel air ratio and reactor throughout. They tried to vary the initial bed height, product gas concentrations, the fuel to air ratio. They concluded that char did not have any effects on performance and acted as a catalyst capable of reducing tar. In the end, all of the computational results did not match with the experimental results.
(Chalermsinsuwan *et al*, 2010) have done CFD modelling of tapered circulating fluidized bed reactor risers. They found out the hydrodynamic descriptions and chemical reaction responses by using Ansys Fluent and k-ε turbulence model. They did not use any extra model or UDF. They tested CFD modelling for two types of risers, namely, taper in and taper out. They found that the tapered-in riser is the best for reactions with a slow rate, while the tapered-out riser is better for the reactions with a fast rate.

2.2 Eulerian-Lagrangian: -

(Zabetta *et al*, 1995) applied CFD to study reduction of NOx by staging in biomass combustion by homogenous detailed chemical kinetic modelling. They solved for k-ε turbulence model of three dimensions and used the Fluent 5.4 software where they used UDF. A staged combustion was done and the results matched with the experimental results. The effect of temperature, devolatilized hydrodynamics, volatile nitrogen components (NH3, HCN) and number of air addition stages on NOx formation was investigated.

(Sofialidis *et al*, 2001) has simulated biomass gasification in fluidized beds using computational fluid dynamics approach. They used the fluent software for simulations along with the k-ε turbulence model but did not use any UDF. Their remarks included that the biomass particle trajectories did not represent the real situation because the effect of the solid-to-solid interaction was ignored.

(Brink *et al*, 2001) continued their previous work and presented a modified approach for predicting NOx emissions trends from biomass fired bubbling fluidized bed boilers. They did CFD modelling of the fate of biomass fuel-nitrogen in the freeboard of a 70 MW Fluidized Bed Combustor and ran simulations on Fluent by using their own UDF. They modelled the effect of turbulence with eddy dissipation model.

At present (Mueller *et al*, 2005) have simulated numerical solutions of the combustion behavior of different biomasses in a bubbling fluidized bed boiler on Ansys Fluent by using k- ε turbulence model and prepared UDF for the 3-D simulation model. They have considered all stages of combustion, namely, drying, devolatilization, char carbon conversion and ash particle formation, to predict the temperature profiles in fluidized bed combustors. They have used CFD to formulate numerical model of peat-forest residue, Refused derived fuel (RDF) respectively in bubbling fluidized bed combustors. To study devolatilization behavior, they have used Kobayashi model.

3. Combustion

Combustion is nothing but the exothermic oxidation process which occurs at high temperature. For a good combustion process, the parameters like optimum time of reaction, appropriate temperature required for reaction to happen and better mixing of fuel with oxidant are mainly required. Fluidized bed combustors are largely used for this purpose. The necessary turbulence required for good combustion is provided with high degree of gas-solid mixing in fluidized bed furnace.

3.1 Physical and chemical processes

The stages during the combustion process can be studied using different CFD models. The first step to initiate the combustion process is heating the fuel particles up to the temperature required for the combustion process. Present CFD codes do not allow for simultaneous modelling of burning particle (E-L approach) and bubbling bed (E-E approach). So, in all the studies it is assumed that the heat required to initiate the combustion of fuel particles is provided by the fluidizing air entering at combustion temperatures. The correct prediction of the temperature is very important for CFD model, as it affects the combustion process. The stages in combustion and gasification process are shown in Fig 3.

3.2 Case Study: -

1) (Agraniotis *et al*, 2009) has not considered all the stages of combustion as they have considered dried fuel entering in combustor.

2) (Mueller *et al*, 2005) and (Wang and Yang *et al*, 2008) have considered all the stages of combustion.

3) (Ravelli *et al*, 2008) used two-mixture fraction approach for correct temperature prediction.

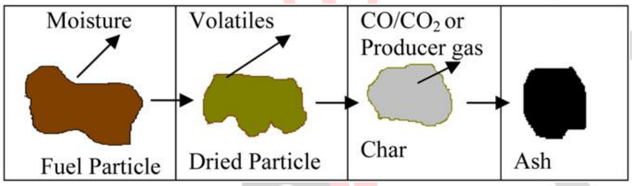

Fig 3- Stages in combustion and gasification process (Singh *et al*, 2013).

3.3 Devolatilization: -

The process in which wide range of gaseous products are released due to decomposition of fuel is known as Devolatilization. Only few studies are available for devolatilization of solid fuel particles in freeboard. Small particles burn faster than big ones hence big particles are expected to be broken into small fragments immediately after feeding, this accelerates the emission of volatile from Refused Derived Fuel (RDF).

3.4 Case Study: -

1) (Mueller *et al*, 2005) and (Raveli *et al*, 2008) have used CFD to formulate numerical model of peat-forest residue and RDF in bubbling fluidized bed combustors in which they have used the Kobayashi model.

2) (Raveli *et al*, 2008) showed the comparison between devolatilization and the char burnout at the entry of the combustors. The study showed that devolatilization was starting after short time when the fuel is injected. The literature reveals that devolatilization depends on fragile structure, size, temperature, density etc. properties of particles.

3.5 Char oxidation/burn out: -

Char burns is nothing but the devolatilized fuel. It is dependent upon intrinsic reactivity and size to burn. Due to computational limitations, the char oxidation and char burnout profiles are very difficult to study. With the present computational approach using Lagrangian approach, it is very easy to study char oxidation in freeboard of commercial as well lab scale units. The char burn-out is difficult to validate in real combustors and it is not validated in any studies. In case of minimum load, char combustion takes place in the region close to furnace exit, while in case of minimum load, it takes place in the area just above the bed.

3.6 Case study: -

1) The study of (Geng and Che *et al*, 2011), (Rong and Horio *et al*, 1999) and (Liu *et al*, 2011) includes DEM-CFD to formulate char combustion model in dense bed.

2) (Rong and Horio *et al*, 1999) used DEM for simulation of the char in bubbling fluidized combustor. They took the fluctuations of char particle temperature into account to affect the particle-particle heat conduction, particle-gas heat convection, radiation and combustion. Their result states that NO emissions are affected by the temperature of burning char particles.

3) (Liu *et al*, 2011) simulated char and propane combustion. They predicted that gaseous fuel reduces the combustion rate. Also, as temperature rises, the char combustion rate in bed decreases.

4) (Geng and Che *et al*, 2011) proposed a DEM-CFD model for the combustion of char in bubbling fluidized bed of inert sand.

3.7 Ash behavior*: -*

Ash deposition is a phenomenon that cannot be neglected. Also, there are many ash related problems such as slagging, fouling and corrosion that cannot be neglected. Ash deposition regions can be found out using CFD modelling. Reliable ash deposition is essential to improve the boiler efficiency. Ash related problems are dependent on factors such as mineral matter distribution in the fuel, agents specific to the used combustion technique as well as design aspects unique for the combustion chamber of any operating power plant. The exact positions of ash particle impact on the boiler surfaces are recorded and the particle temperatures at these locations are the linking parameter to the fuel specific stickiness criterion. Reliable data for the ash melting behavior is obtained from chemical fractionation and the thermodynamic equilibrium calculations. The higher temperature regions in furnace are found with the help of CFD. As shown in the Fig 4, different colors are associated with different temperature regions where the hot regions with red in color are responsible for ash prediction in fluidized bed furnace. But the basic methodology of ash prediction using CFD remains same in all the studies reported.

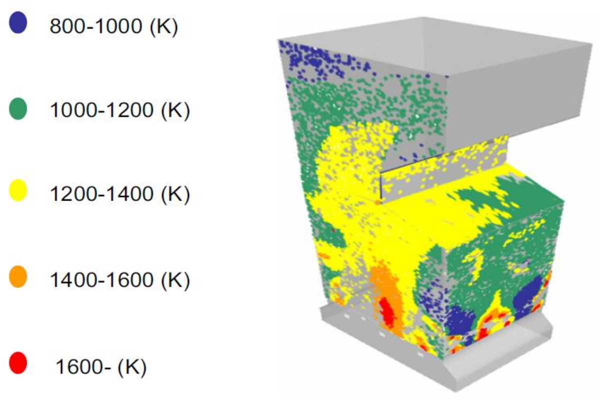

Fig 4- Visual validation of ash deposit prediction in the freeboard of a bubbling fluidized bed furnace (Singh *et al*, 2013).

3.8 Particle trajectories: -

Lagrangian formulation is used by CFD model to compute particle trajectory. Lagrangian formulation includes inertia, hydrodynamic drag, force of gravity etc. In freeboard, the particle concentration is quite less compared to gas, whereas, in the dense bed, the particle concentration is quite more compared to gas. Therefore, it is not possible to track the particle in the dense bed, hence particle trajectories are only possible in freeboard. Particle trajectory is dependent upon size, density and moisture content of the particle. In the last stage, the ash particles are ejected out from the furnace.

3.9 Fuels in gaseous phase: -

Gaseous phase eliminates the heterogeneous reactions. As this system uses less computational space, it is easier to study. There are very few number of research papers which are focused on combustion of gaseous fuels in fluidized bed. Of which, few have discussed combustion of gaseous fuels in fluidized bed using CFD. Most of these studies are on 2D geometries. Many technical challenges are needed to be overcome to use this model to large scale industrial processes. CLC (chemical looping combustion) typically employs a dual fluidized bed system. In dual fluidized bed system, a metal oxide is employed as a bed material providing the oxygen for combustion in the fuel reactor. Further, the reduced metal is transferred to the second bed (air reactor) and it is further re-oxidized before it is reintroduced back to the fuel reactor completing the loop. Solid fuels are addressable and good mixing can be done, therefore the current systems are mostly based on fluidized bed technology. Intensive research has been performed from past few years involving chemical looping combustion but it is still much away from being commercially available technology. So, there is lot of scope in the study of this model. CFD modelling of chemical loop combustion system has been done to know how the fuel and the oxygen carrier is circulated in both vessels along with the other parameters. There are many parameters apart from time, temperature and turbulence such as particle diameter, flow rate, superficial velocity, design parameters, oxygen concentration etc. which affect combustion process. Huge literature is available on these parameters but, only few studies with CFD are reported.

Case Study: -

1) In the study of (Emden *et al*, 2010), development and testing of an interconnected multiphase CFD model for chemical looping combustion is done.

2) (Shuai *et al*, 2011) performed CFD simulations of fuel reactor using chemical looping combustion process. High weight fraction of unburnt methane fuel in the flue gas along with CO_2 and H_2O is found in the fuel of the fuel reactor during the simulation. This implies high fuel loss at the exit of the reactor.

3)(Deng *et al*, 2008, 2009, 2009) used multiphase CFD modelling for chemical looping combustion process. In this work, the reaction kinetic models of $CaSO_4$+H_2O fuel reactors are developed.

4) Under the study of (Mahalatkar *et al*, 2011), the simulations of a circulating fluidized bed chemical looping combustion system utilizing gaseous fuel are made. Methane is used as fuel during simulation along with a 2D continuum model to describe both the phases i.e., gas and solid phase.

5) (Jung and Gamwo *et al*, 2008) applied multiphase CFD based models for chemical looping combustion process using MFIX code.

4. Gasification: -

Gasification is a process that converts organic or fossil fuel-based materials into CO_2 and syngas. It is achieved by a chemical reaction of a limited amount of oxygen with fuel such as coal, heavy oil or petroleum coke. Sometimes, we also provide steam for better chemical reactions. Here CFD is used in studying the process of gasification.

4.1 Physical and chemical processes-

E-E TFM CFD model is used to study the physical and chemical behavior of fuels in fluidized bed gasification process. Here we studied about the gasification of coal as fuel. (Armstrong *et al*, 2010) formed a CFD model for biomass and coal. They included the coal to the model to see the effect on the gaseous products. (Zeng *et al*, 2008) used coal in fluidized bed and studied the effects using CFD model. Increase in pressure causes the increase in bed temperature, due to which rate of Boudouard reaction increases and production of CO is increased. (Yu *et al*, 2007) formulated the two-dimensional model for bubbling fluidized bed gasifier (BFBG). Advantages of two-dimensional model over 2-

phase or 3-phase one-dimensional model are explained under this model. Using 2-D model, the simulation results obtained will be more exact for predicting the distribution of pressure, temperature, velocity, volume fraction of the phases and gas composition. (Cornejo and Farias *et al*, 2011) formulated the same model but with more simplifications. The major difference in this model from (Yu *et al* 2007) is that, in this process, coal enters in dried state in gasifier and ash is not considered. Now, studying about the physical properties of Biomass (char 1), Coal (char 2), and Limestone, graphs are plotted for the volume fractions. In Fig 5(a), we see the comparison between volume fraction of Biomass (char 1) and Coal (char 2). We know that Biomass have higher density than coal and also the diameter of particles of Biomass are larger than coal. So, Biomass settles down to the base of bed as shown in Fig 5(b). In Fig 5(c) and Fig 5(d), we see the comparison between volume fraction of Limestone and char. We know that char have lower density so these char particles flow and segregate to the top of the bed.

4.2 Effect of Operational Parameters: -

CFD are used to study the effect of operational parameters like particle size, flow velocity, concentration, bed materials, etc. on the gasification process in fluidized bed. (Gerber *et al*, 2010) formed the TFM E-E CFD model for char as bed material, varied the initial bed height by 25% in both directions. This shows the influence only on the reactive tar component and not on the gaseous components. (Deng *et al*, 2008) in his model, varies the bed temperature and pressure. Increase in pressure results in increase in gas quality. (Armstrong *et al*, 2011) in his study of BFB gasifier using CFD, increased the bed temperature which resulted in faster consumption of emitting gases like H_2O and CO_2, which in result decreased the amount of H_2O and CO_2.

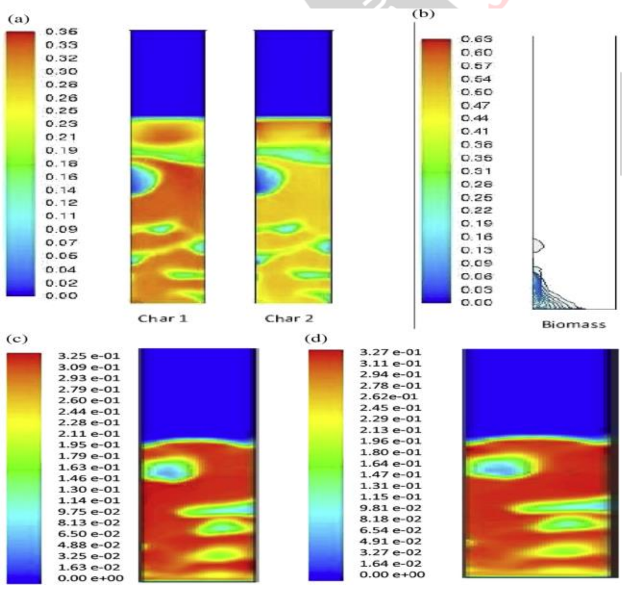

Fig 5-(a) Volume fraction of char1 and char 2 (Armstrong *et al*, 2010). (b) Volumetric fraction of biomass (Armstrong *et al*, 2010). (c) and (d) Volume fraction of lime and char (Armstrong *et al*, 2011).

4.3 Bed Material: -

CFD is used to study the effect of bed material on gasification process. If the bed material is inert then there is no effect on gasification process in fluidized bed. Many authors have used char as bed material to study gasification in fluidized bed. They didn't consider ash phase i.e., the phase in which the fuel will break down into char and combustible gases. Many authors also considered mixture of sand, fuel, char or ashes as a bed material. But there is no known effect of materials on gasification in fluidized bed.

4.4 Design: -

Authors made many variations in design of circulating fluidized bed reactor risers like using tapered in and tapered out. They found that tapered in riser gives more uniform temperature distribution as compared to tapered out. Tapered in is best in slow rate reactions while tapered out is best in fast rate reactions. CFD is helpful in design of fluidized bed devices and is used to study the effect of operational parameters.

5. Conclusion:

1. CFD uses numerical analysis and data structures to solve and analyze the combustion and gasification of fuels in fluidized bed devices.
2. It is used to predict the behavior of fuel in gasification process during combustion in fluidized bed.
3. It is a powerful tool used to study the effect of bed temperature, heat flow, ash, emission products during combustion and gasification process in fluidized bed.
4. Study of physical and chemical processes and effects of operational parameters could affect the performance of devices.
5. E-E TFM CFD model do not have any effect on design of industrial fluidized bed units in the process.
6. CFD simulations have many approximate models as well as assumptions but still results obtained from CFD simulations are much accurate with experimental data.
7. On varying the size of biomass particles, the Eulerian-Eulerian approach in dense fluidized bed is not possible until wide approximations are made.
8. So, CFD modelling using Eulerian-Eulerian approach still needs to be explored.
9. The aspects like mixing of fuel in dense bed, ash deposition, fuel combustion behavior during feeding, etc. with CFD still needs to be explored.
10. Still, there are some obstacles like inaccuracy in simulating large 3D problems on an affordable computer but, with the progression of the computing power and the development

of chemical and physical models, CFD applications will spread more widely in future.

6. References

Pan T.W., Joseph D.D., Bai R. Glowinski, R. Sarin V., (2002), Fluidization of 1204 spheres: simulation andexperiments, *Journal of Fluid Mechanics*, 451, pp 169.

Anderson T.B., Jackson R., (1967), A fluid mechanical description of fluidized beds, Equations of motion, *Industrial & Engineering Chemistry Fundamentals,* 6, pp 527.

Drew D.A., (1971), Averaged field equations for two-phase media, *Studies in Applied Mathematics,* 50, pp 133.

Drew D.A., (1983), Mathematical modelling of two-phase flow, *Annual Review of Fluid Mechanics*, 15, pp 261.

Drew D.A., Lahey R.T., (1993), Analytical Modelling of Multiphase Flow, *Particulate Two-Phase Flow*, vol. 510, Butterworth-Heinemann, Stoneham, USA.

Drew D.A., Segel L.A., (1971), Averaged equations for two-phase flows, *Studies in Applied Mathematics,* 50, pp 205.

Enwald H., Peirano E., Almstedt A.E., (1996), Eulerian two-phase flow theory applied to fluidization, *International Journal of Multiphase Flow,* 22, pp 21.

Jackson R., (1997), Locally averaged equations of motion for a mixture of identical spherical particles and a Newtonian fluid, *Chemical Engineering Science*, 52, pp 2457.

Jackson R., (1998), Erratum, *Chemical Engineering Science,* 53, pp 1955.

Jackson R., (2000), The Dynamics of Fluidized Particles, *Cambridge Monographs on Mechanics,* Cambridge University Press, Cambridge, UK.

Whitaker S., (1969), Advances in the theory of fluid motion in porous media, *Industrial and Engineering Chemistry,* 61, pp 14.

Zhang D.Z., Prosperetti A., (1994), Averaged equations for inviscid disperse two-phase flow, *Journal of Fluid Mechanics,* 267, pp 185.

W. Zhou, C.S. Zhao, L. Duan, C.R. Qu, X.P. Chen (2011), Two-dimensional computational fluid dynamics simulation of coal combustion in a circulating fluidized bed combustor, *Chemical Engineering Journal*, 166 (1), pp 306-314.

W. Zhou, C.S. Zhao, L. Duan, D. Liu, X.P. Chen (2011), CFD modeling of oxy-coal combustion in circulating fluidized bed, *International Journal of Greenhouse Gas Control*, 5 (6), pp 1489-1497.

W Zhou, C S Zhao, L Duan, C R Qu, J Y Lu, X P Chen (2009), Numerical simulation on hydrodynamics and combustion in a circulating fluidized bed under O2/CO2 and air atmospheres, in: *Proceeding of 20th International Conference on Fluidized bed combustion*, pp 883-888.

Nikolopoulos I., Rampidis, N. Nikolopoulos, P. Grammelis, E. Kakaras (2009), Numerical investigation of 3- D transient combusting flow in a 1.2 MW_{th} pilot power plant, in: *Proceeding of 20th International Conference on Fluidized bed combustion*, pp 839-844.

K. Myöhänen, T. Hyppänen, A. Vepsäläinen (2006), Modelling of circulating fluidized bed combustion with a semi-empirical three-dimensional model, SIMS 2006, in: *Proceedings of the 47th Conference on Simulation and Modelling*, pp 194-199.

A Brink, S Bostrom, P Kilpinen and M Hupa (2001), CFD modeling of the fate of biomass fuel-nitrogen in the freeboard of a 70 MW FBC-new aspects on controlling mechanisms, in: *First biennial meeting, The Scandinavian-Nordic Section of the Combustion Institute*, Chalmers University of Technology, Goteborg, Sweden.

A. Brink, C. Mueller, M. Hupa, A modified approach for predicting NOx emission trends from biomass fired bubbling fluidized bed boilers, in: *Proceeding of 19th FBC conference*, May 21-24, 2006.

A. Brink, M. Hupa, E. Kurkela, M. Suomalainen (2005), Minimizing NOx emission from a waste derived fuel gasifier gas combustor using CFD combined with detailed chemistry, *IFRF Combustion Journal*. ISSN: 1562-479X 05. ISSN: 1562-479X 1-14.

R. Zevenhoven, M. Järvinen (2001), Particle/turbulence interactions, mass transfer and gas/solid chemistry in a CFBC riser, *Flow Turbulence and Combustion,* 67 (2), pp 107-124.

C. Mueller, A. Brink, M. Hupa (2005), Numerical simulation of the combustion behavior of different biomasses in a bubbling fluidized bed boiler, in: *Proceedings of 18th International conference on Fluidized bed combustion*, Toronto, Ontario, Canada.

C. Mueller, B.J. Skrifvars, R. Backman, S. Nickull, M. Hupa (2001), Simulation of ash particle behavior on surfaces of biomass fired fluidized bed boilers combination of computational fluid dynamics and advanced ash analysis, in: *Proceedings of 16th International conference on Fluidized bed combustion*, Reno, Nevada USA, Sustainable Fuels-Biomass, Paper No. 139.

C. Mueller, B.J. Skrifvars, R. Backman, M. Hupa (2003), Ash deposition prediction in biomass fired fluidized bed boilers-combination of CFD and advanced fuel analysis, *Progress in Computational Fluid dynamics*, 3 (2-4), pp 112-120.

C. Mueller, D. Lundmark, B.J. Skrifvars, M. Hupa (2002), CFD based ash deposition prediction in a bubbling fluidised bed combustor firing mixtures of biomasses, in: *Proceeding Finnish-Swedish Flame Days*, Vassa, Finland.

D. Lundmark, C. Mueller, B.J. Skrifvars, M. Hupa (2007), Computational fluid dynamic modeling of combustion and ash deposition in a biomass co-fired bubbling fluidized bed boiler, *International Journal Energy Cleaner Environment,* 8 (2).

D. Lundmark, C. Mueller, R. Backman, M. Zevenhoven, B.J. Skrifvars, M. Hupa (2010), CFD based ash deposition prediction in a BFBC firing mixtures of peat and forest residue, *Journal of Energy Resources Technology*, Transactions of the ASME, 132 (3).

S. Ravelli, A. Perdichizzi, G. Barigozzi (2008), Description, applications and numerical modeling of bubbling fluidized bed combustion in waste-to-energy plants, *Progress in Energy and Combustion Science,* 34, pp 224-253.

M. Rozainee, S.P. Ngo, A.A. Salema, K.G. Tan (2010), Computational fluid dynamics modeling of rice husk combustion in a fluidized bed combustor, *Powder Technology,* 203, pp 331-347.

M. Agraniotis, D. Stamatis, P. Grammelisand Kakaras (2009), Numerical investigation on the combustion behavior of pre-dried Greek lignite, *Fuel,* 88 (12), pp 2385-2391.

A. Brink, O. Karlström, M. Hupa (2009), A simplified model for the behavior of large biomass particles in the splashing zone of a bubbling bed, in: *Proceeding of 20th International Conference on Fluidized bed combustion*, pp. 764-767.

W. Yuxin, L.V. Junfu, J. Zhang, L. Qing, Yue Guangxi, Y. Zhang, Y. Long, Z. Yang (2004), Conceptual design of an 800 MW supercritical pressure circulating fluidized bed boiler, *Boiler Technology*,3(48), pp 1-5.

W.M. Gao, L.X. Kong, P.D. Hodgson (2007), Computational simulation of gas flow and heat transfer near an immersed object in fluidized beds, *Advances in Engineering Software*, 38, pp 826-834.

Y. Behjat, S. Shahhosseini, S.H. Hashemabadi (2008), CFD modeling of hydrodynamic and heat transfer in fluidized bed reactors, *International Communications in Heat and Mass Transfer,* 35, pp 357-368.

R. I. Singh, A. Brink, M. Hupa (2013), CFD modelling to study fluidized bed combustion and gasification, *Applied Thermal Engineering,* 52, pp 585-614.

N.V. Gnanapragasam, B.V. Reddy (2009), Numerical modeling of axial bed-to-wall heat transfer in a circulating fluidized bed combustor, *International Journal of Heat and Mass Transfer,* 52, pp 1657-1666.

W. Wang, B. Lu, N. Zhang, Z. Shi, J. Li (2010), A review of multiscale CFD for gas-solid CFB modeling, *International Journal of Multiphase Flow,* 36, pp 109-118.

X.Z. Chen, D.P. Shi, X. Gao, Z.H. Luo (2011), A fundamental CFD study of the gas-solid flow field in fluidized bed polymerization reactors, *Powder Technology*, 205, pp 276-278.

N.G. Deen, M.V. Annaland, M.A. Van der Hoef, J.A.M. Kuipers (2007), Numerical simulation of dense gas-solid fluidized beds: A multiscale modeling strategy, *Chemical Engineering Science,* 62, pp 28-44.

L. Yu, J. Lu, X. Zhang, S. Zhang (2007), Numerical simulation of the bubbling fluidized bed coal gasification by the kinetic theory of granular flow (KTGF), *Fuel*, 86, pp 722-734.

J. Jung, I.K. Gamwo (2008), Multiphase CFD-based models for chemical looping combustion process: fuel reactor modeling, *Powder Technology*, 183 (3), pp 401-409.

Z. Deng, R. Xiao, B. Jin, H. Huang, L. Shen, Q. Song, L. Qianjun (2008), Computational fluid dynamics modeling of coal gasification in a pressurized spout-fluid bed, *Energy and Fuels*, 22, pp 1560-1569.

Z. Deng, R. Xiao, B. Jin, Q. Song, H. Huang (2008), Multiphase CFD modeling for a chemical looping combustion process (fuel reactor), *Chemical Engineering Technology*, 31 (12), pp 1754-1766.

H. Kruggel-Emden, S. Rickelt, F. Stepanek, A. Munjiza (2010), Development and testing of an interconnected multiphase CFD-model for chemical looping combustion, *Chemical Engineering Science*, 65, pp 4732-4745.

S. Gerber, F. Behrendt, M. Overmann (2010), An Eulerian modeling approach of wood gasification in a bubbling fluidized bed reactor using char as bed material, *Fuel*, 89, pp 2903-2917.

B. Chalermsinsuwan, P. Kuchonthar, P. Piumsomboon (2010), CFD modeling of tapered circulating fluidized bed reactor risers: hydrodynamic descriptions and chemical reaction responses, *Chemical Engineering and Processing,* 49, pp 1144-1160.

E.C. Zabetta, T. Norstrom, P. Kilpinen, M. Hupa (1999), NOx Reduction by Staging in Biomass Combustion - A Kinetic and CFD Modelling Study, *The Joint Swedish Finnish Flame Day*, IFRF, Vexjo, Sweden, pp 1-17.

D. Sofialidis, O. Faltsi (2001), Simulation of biomass gasification in fluidized beds using computational fluid dynamics approach*, Thermal Science*, 5 (2), pp 95-105.

L.M. Armstrong, S. Gu, K.H. Luo (2011), Parametric study of gasification processes in a BFB coal gasifier, *Industrial and Engineering Chemistry Research,* 50, pp 5959-5974.

A Brink, S Bostrom, P Kilpinen and M Hupa (2001), CFD modeling of the fate of biomass fuel-nitrogen in the freeboard of a 70 MW FBC-new aspects on controlling mechanisms, in: *First biennial meeting, The Scandinavian-Nordic Section of the Combustion Institute*, Chalmers University of Technology, Goteborg, Sweden.

C. Mueller, A. Brink, M. Hupa (2005), Numerical simulation of the combustion behaviour of different biomasses in a bubbling fluidized bed boiler, in: *Proceedings of 18th International conference on Fluidized bed combustion*, Toronto, Ontario, Canada.

S. Ravelli, A. Perdichizzi, G. Barigozzi (2008), Description, applications and numerical modeling of bubbling fluidized bed combustion in waste-to-energy plants, *Progress in Energy and Combustion Science*, 34, pp 224-253.

Y. Wang, L. Yan (2009), CFD based combustion model for sewage sludge gasification in a fluidized bed, *Frontiers of Chemical Engineering in China*, 3 (2), pp 138-145.

Z.S. Yu, X.Q. Ma, Z.Y. Lai, H.M. Xiao (2009), CFD modelling applied to the co-combustion of paper sludge and coal in a 130 t/h CFB boiler, in: *Proceeding of 20th International conference on fluidized bed combustion,* pp 1165-1170.

Energy Efficiency in Thermal Utilities. A Guide Book for Energy Managers and Auditors, Bureau of Energy Efficiency, Ministry of Power, Government of India, 2005.