Numerical Analysis of Full Loop Model Circulating Fluidized Bed Combustion Boiler

Nandhini.M[1],Dr.Ramesh.K[2]
[1]PG Student, Department of Mechanical Engineering, Government College of Technology, Coimbatore
[2]Associate Professor, Department of Mechanical Engineering, Government College of Technology, Coimbatore

Abstract- *The main aim of this project is to study the hydrodynamics parameters like velocity and pressure variations, fluidization parameters like minimum fluidization velocity, expansion ratio, and fluctuation ratio of full loop circulating fluid bed combustion boilers model by combining with cyclone separator and seal loop for different bed heights by using different type of nozzles.*

And the numerical calculation and flow characteristics will be done through ANSYS fluent Multiphase Software. Finally for better optimization the results obtained from the numerical calculation will be compared.

I. INTRODUCTION

Fluidized bed boilers are suitable for larger scale heat plants, and well established at the scale of 100 MW, although installations with an output of 2 MW exist. In fluidized bed combustion, rapid mixing ensures uniformity of temperature. The main advantage of fluidized bed combustion system is that municipal waste, sewage plant sludge, biomass, agricultural waste and other high moisture fuels can be used for heat generation.CFB technology burns fuels without fire or burners in the furnace but through a process of fluidization mixes the fuel particles with limestone which captures the Sulphur oxides that are formed and the low temperature reduces nitrogen oxide formation. The limestone and fuel are recycles multiples times which increases both the efficiency of producing high quality steam to produce power and also reduces pollution due to its ability to burn fuel in a clean way than the conventional processes. Circulating fluidized bed technology (CFB) also offers the flexibility of using both coal as well as biomass fuels.Air supply is from under the bed at high pressure. This lifts the bed material and the coal particles and keeps it in suspension. The coal combustion takes place in this suspended condition. This is the Fluidized bed. Special design of the air nozzles at the bottom of the bed allows air flow without clogging. Primary air fans provide the preheated Fluidizing air. Secondary air fans provide pre-heated Combustion air. Nozzles in the furnace walls at various levels distribute the Combustion air in the furnace.Fine particles of partly burned coal, ash and bed material are carried along with the flue gases to the upper areas of the furnace and then into a cyclone. In the cyclone the heavier particles separate from the gas and falls to the hopper of the cyclone. This returns to the furnace for re circulation. Hence the name Circulating Fluidized Bed combustion. The hot gases from the cyclone pass to the heat transfer surfaces and go out of the boiler. CFBC boilers are said to achieve better calcium to sulphur utilization – 1.5 to 1 vs. 3.2 to 1 for the AFBC boilers, although the furnace temperatures are almost the same. CFBC requires huge mechanical cyclones to capture and recycle the large amount of bed material, which requires a tall boiler.

Adnan Almuttahar et al.[1] studied the hydrodynamics of gas–solid flow in a circulating fluidized bed (CFB) riser at various fluidization conditions using the Eulerian–Granular multiphase model. The model was evaluated comprehensively by comparing its predictions with experimental results reported for a CFB riser operating at various solid mass fluxes and superficial gas velocities. The model was capable of predicting the main features of the complex gas–solids flow, including the cluster formation of the solid phase along the walls, for different operating conditions. The model also predicted the coexistence of up flow in the lower regions and downward flow in the upper regions at the wall of the riser for high gas velocity and solid mass flux, as reported in the literature. Two fluid modelling (TFM) to a two dimensional and three dimensional circulating fluidized bed (CFB) was studied by Armstrong et al.[2] An energy minimization multi scale (EMMS) based drag model is compared with a classical drag model, namely the Gidaspow model in the light of experimental data from the CFB. The axial particle velocities and the radial volume fraction at different heights are considered. The specularity coefficient responsible for the tangential solid velocities at the walls is varied to study the effect on the down flow of particles at the wall. The work is further extended to explore the effects of velocity variation on the flow distribution showing the transition from a bubbling to a fast fluidizing regime. Furthermore, the diameters of the bubbles observed within the bubbling regime are compared with the Davidson’s bubble diameter model for a range of particle diameters. Cenfan Liu et al.[3] had done their experiment through on-line adjustment of solids flow rate via a mechanical valve in CFBC boiler by simulation. The two fluid model (TFM) is used as the governing equations, for which the solids stress is closed by using the kinetic theory of granular flow and the drag is closed by using the EMMS/matrix scheme Kun Luo et al.[4] studied in the full loop gas–solid motions in a three dimensional circulating fluidized bed are numerically modeled using the computational fluid dynamics combined with the discrete element method. The time averaged flow characteristics and the particle scale details related to solid motion are discussed. Three dimensional (3D), time dependent simulation of a full loop CFB revealed that the axial profiles of cross sectionally averaged solid volume fraction, and the radial profiles of solid axial velocity and solid volume fraction in a Eulerian granular multiphase model with a drag coefficient correction based on the energy minimization multi-scale (EMMS) model which was used to simulate a semi industry scale circulating fluidized bed (CFB) by Nan Zhang et al.[5].Wojciech et al.[6] studied a hybrid Euler–Lagrange model known as the dense discrete phase model (DDPM), which has common roots with the multiphase particle in cell model, was applied in simulating particle transport within a mid-sized experimental CFB facility. Implementation of the DDPM into the commercial ANSYS Fluent CFD package is relatively young in comparison with the granular Eulerian model. For that reason, validation of the DDPM approach against experimental data is still required and is addressed in this paper.

II. GEOMETRY AND OPERATING CONDITION

Computational results are obtained by analyzing a CFD model (with similar dimensions of experimental model) in ANSYS multiphase.

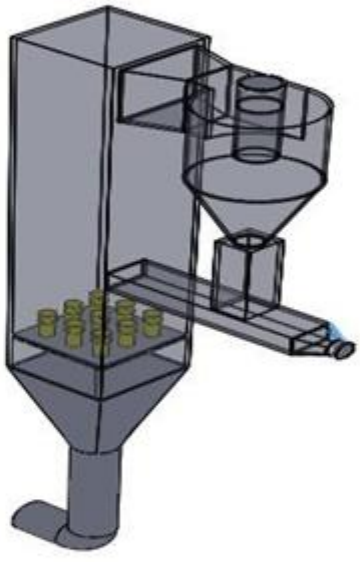

Fig 1. Modelled design of full loop CFBC boiler

Input conditions

Multiphase- ON- Eulerian- Implicit Phase 1- Air- Density = 1.225kg/m3 Phase 2- Sand- Density = 2500kg/m3 Inlet condition:
Values for primary velocity = 1 m/s, 2 m/s, 3ms Secondary velocity = 2 m/s
Outlet condition: Pressure = 1 bar

Values for defining the sand bed: X_{min} = -0.1m, X_{max} = 0.1 m,

Y_{min} = 0.106m, Y_{max} = 0.256m Z_{min} = -0.1m, Z_{max} = 0.1m

III. MODELING EQUATIONS

Calculation of velocity of air at nozzle inlet:

The velocity at the nozzle inlet varies from the outlet velocity of blower due to the change in area. Hence, for various velocities of air from the blower, the velocity at the nozzle inlet has to be found. This can be obtained by determining the flow rate, Q. Flow rate,Q = A x V
For blower outlet, Area = (π/4) x D2

= (π/4) x 0.0752

Area of blower outlet = 0.004416 m2 Therefore,
Flow rate (for maximum velocity of air from blower), Q = 0.004416 x 30

Q = 0.132469 m3/s

Similarly,

Flow rate (for minimum velocity of air from blower), Q = 0.004416 x 5

Q = 0.022078 m3/s

We know that flow rate remains same in blower outlet and nozzle inlet. Therefore,

Velocity at nozzle inlet for the maximum velocity of blower = 0.132469 / (0.2 x 0.20)

Vmax = 3 m/s

Velocity at nozzle inlet for the minimum velocity of blower = 0.022078 / (0.2 x 0.2)

Vmin = 0.5m/s

Calculation of terminal velocity and minimum fluidization velocity:

Density of air, ρg = 1.1649 kg/m3 Density of sand, ρp = 2500 kg/m3
Dynamic viscosity of air, μ = 1.1649 x 10-5 x (ρp – ρg)

= 2500 – 1.649

= 2498.835 Kg/m3

Archimedes number, Ar = [ρg* (ρp - ρg)*g*dp3]*μ2 Substituting the values, we get Ar = 2209.557311

Reynolds number at minimum fluidization condition Remf = $[C_1 + C_2 \text{ Ar}]^{0.5} – C_1$ C_1 and C_2 are empirical constant whose values are 27.2 and 0.0408 respectively. Substituting the values, we get Remf = 1.609

Minimum fluidization velocity, Umf = (μ x Remf) / (dp x dg)

Substituting the values, we get Umf = 0.08m/s Terminal velocity, Ut = [μ/(dp*dg)]*(Ar/7.5)0.666 Substituting the values, we get Ut = 2.35m/s

Calculation of expansion ratio and fluctuation ratio:

Expansion ratio can be defined as the ratio of maximum bed height to static bed height during fluidization.

Expansion ratio of bubble cap nozzle at 150 mm bed height for minimum velocity
=200/150

Expansion ratio = 1.333

Expansion ratio of bubble cap nozzle at 150 mm bed height for minimum velocity

= 380/150

Expansion ratio = 2.533

Fluctuation ratio can be defined as the ratio of maximum bed height to minimum bed height during fluidization.

Fluctuation ratio of bubble cap nozzle at 150 mm bed height for minimum velocity

= 200/160

Fluctuation ratio = 1.25

Fluctuation ratio of bubble cap nozzle at 150 mm bed height for minimum velocity

= 380/250

Fluctuation ratio = 1.52

IV. SIMULATION RESULTS

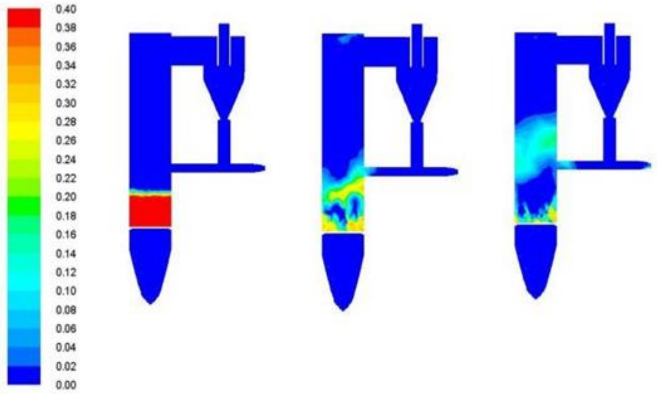

Fig.2: Volume fraction contour while using bubble cap nozzle

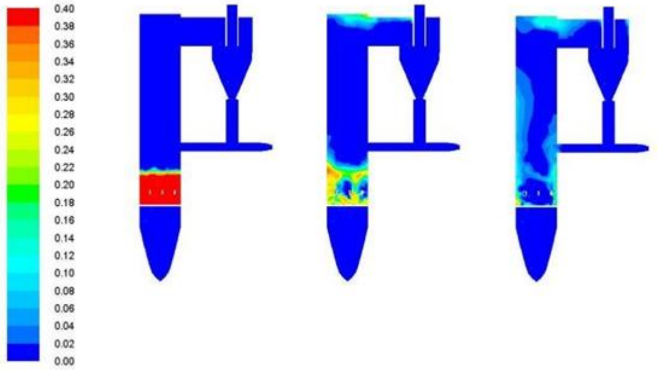

Fig.3: Volume fraction contour while using modified arrowhead nozzle

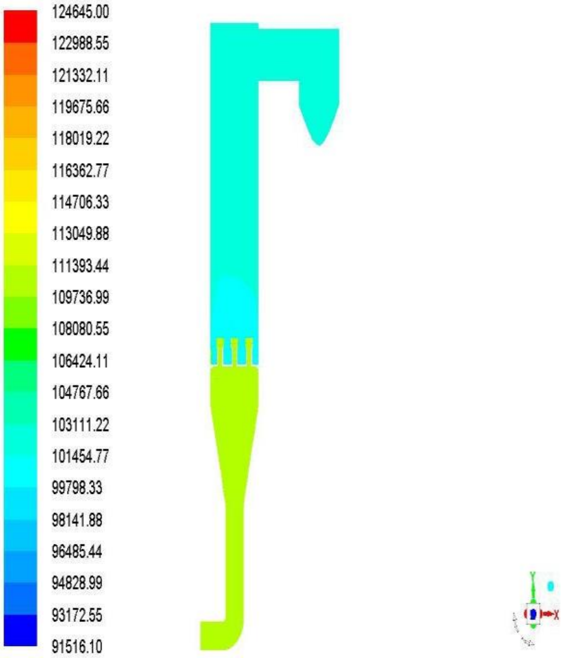

Fig.4: Absolute pressure contour for bubble cap nozzle

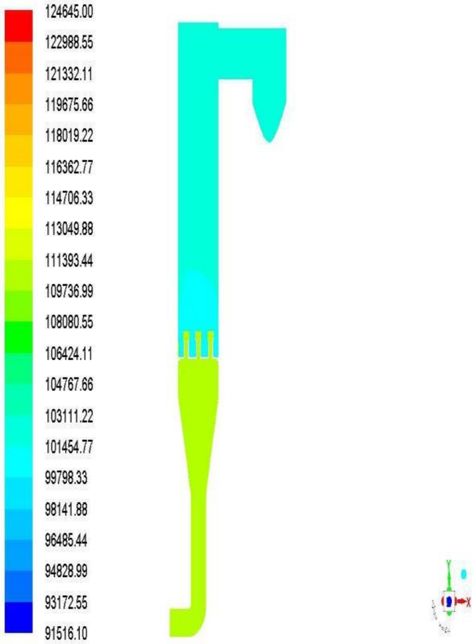

Fig.5: Absolute pressure contour for modified arrow head nozzle

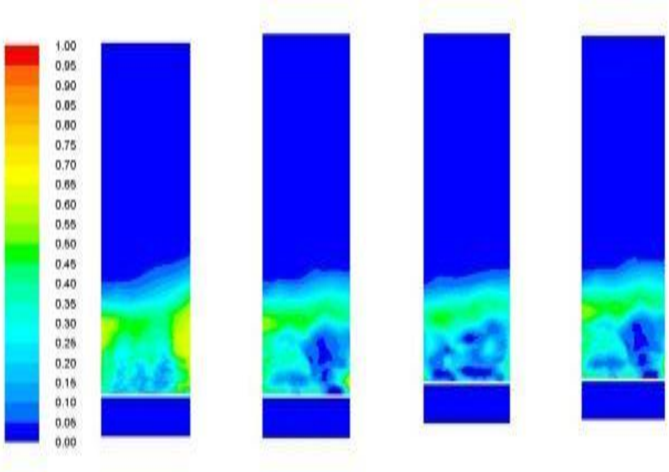

Fig. 6: Similarity in bed movement while using modified arrowhead nozzle

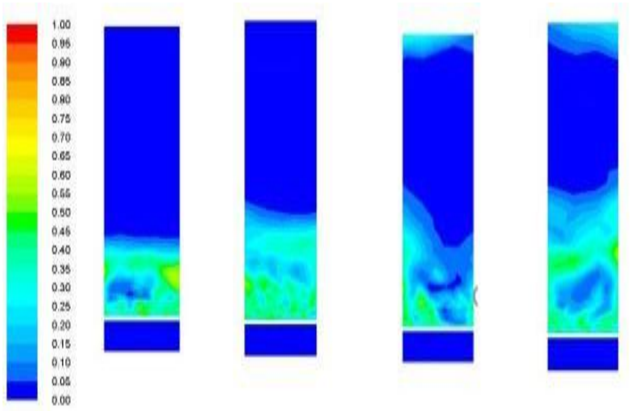

Fig. 7: Similarity in bed movement while using bubble cap nozzle

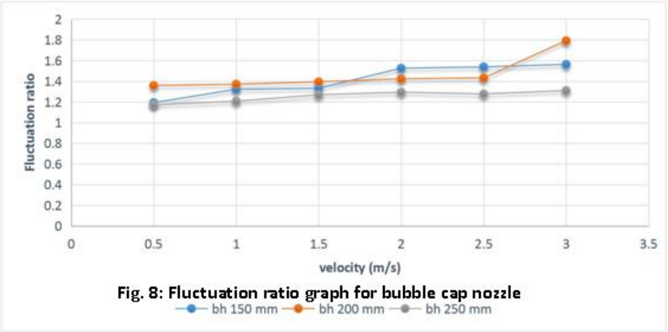

Fig. 8: Fluctuation ratio graph for bubble cap nozzle

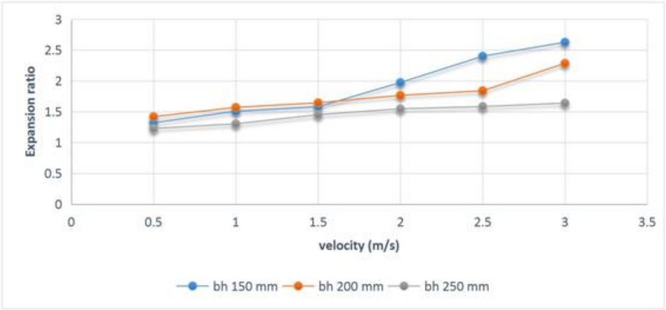

Fig. 9: Expansion ratio graph for bubble cap nozzle

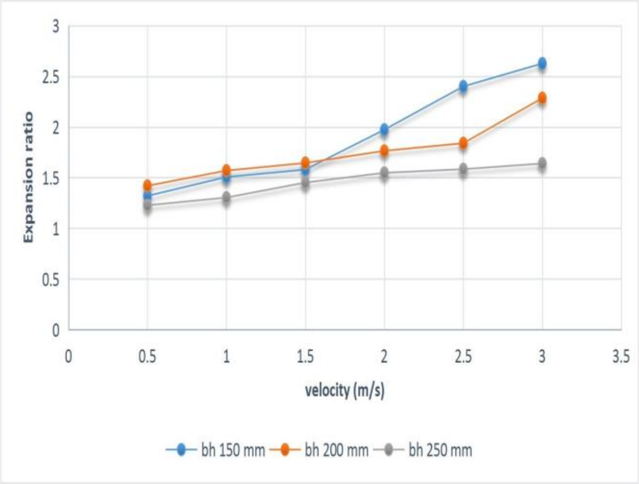

Fig. 10: Expansion ratio graph for modified arrow head nozzle

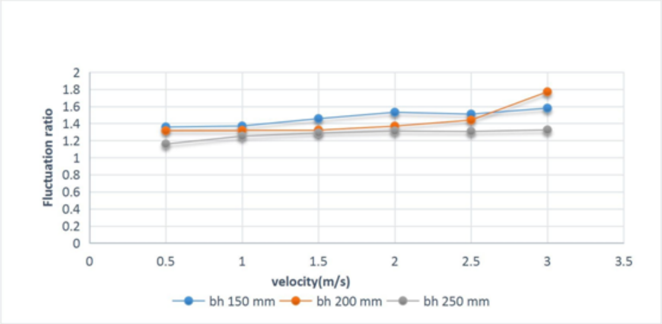

Fig. 11 : Fluctuation ratio graph for modified arrow head nozzle

The variation in fluctuation ratio at various velocity at various bed height of 150 mm, 200 mm, 250 mm while using bubble cap nozzle can be illustrated by Fig. 8.The variation in expansion ratio at various velocity at various bed height of 150 mm, 200 mm, 250 mm while using bubble cap nozzle can be illustrated by Fig. 9.The variation in fluctuation ratio at various velocity at various bed height of 150 mm, 200 mm, 250 mm while using modified arrowhead nozzle can be illustrated by Fig. 10.The variation in expansion ratio at various velocity at various bed height of 150 mm, 200 mm, 250 mm while using modified arrowhead nozzle can be illustrated by Fig.11.The pressure drop value obtained from CFD analysis for bubble cap nozzle is 3.70KPa. The pressure drop value obtained from CFD analysis for modified arrow head nozzle is 3.260KPa.On comparing the results obtained from the numerical analysis, it can be clearly said that there is huge parallelism in both the for the expansion ratio, fluctuation ratio and for the bed movement. Both the expansion ratio, fluctuation ratio is increases with increases in velocity for each bed height.The difference in the expansion ratio and fluctuation ratio value due to some practical error.

V. CONCLUSION

Based on the CFD simulation the following conclusions were arrived:

- The expansion ratio and fluctuation ratio for 150 mm bed height is greater than the other two bed height.
- Pressure drop for modified arrow head nozzle is less compared with bubble head nozzle.
- Both the expansion ratio and fluctuation ratio is increases with the increase in the velocity at various bed height.
- Fluidization index is good for bubble cap when compared to modified arrow head.
- The expansion ratio and fluctuation ratio for bubble cap is higher when compared to modified arrow head.
- Modified arrow head prevents the back flow of sand to air duct.
- Bubble diameter are small in bubble cap than the modified arrow head nozzles.
- Turbulence are well created in bubble cap than the modified arrow head nozzles.

REFERENCES

[1] Adnan Almuttahar, Fariborz Taghipour (2007), ‘Computational fluid dynamics of a circulating fluidized bed under various fluidization conditions’, University of British, Columbia, Chemical Engineering Science, Vol 63, pp 1696 – 1709.

[2] Armstrong L.M, K.H. Luo, S. Gu (2010), ‘Two-dimensional and three-dimensional computational studies of hydrodynamics in the transition from bubbling to circulating fluidised bed’, University of Southampton, United Kingdom, Chemical Engineering Journal, Vol 160 , pp 239–248.

[3] Cenfan Liu, Ming zhao Zhao, Wei Wang, Jinghai Li, (2015) ‘3D CFD simulation of a circulating fluidized BEd with on-line adjustment of mechanical valve’, University of Chinese Academy of Sciences, Beijing 100049, China, Chem. Eng. Science, Vol 53, pp 3367-3379.

[4] Kun Luo, Fan Wu, Shiliang Yang, Mingming Fang, Jianren Fan (2014) ‘High-fidelity simulation of the 3-D full-loop gas–solid flow characteristics in the circulating fluidized bed’, State Key Laboratory of Clean Energy Utilization, Zhejiang University, Hangzhou 310027, PR China, Chemical Engineering Science, Vol 123, pp 22–38.

[5] Nan Zhang, Bona Lu, Wei Wang, Jing Hai Li, (2008), 'Virtual experimentation through 3D full-loop simulation of a circulating fluidized bed', Chinese Academy of Sciences, Beijing 100190, China, Particuology, Vol 6, pp 529–539.

[6] Wojciech P. Adamczyk, Adam Klimanek, Ryszard A. Białecki, Gabriel Wecel,Paweł Kozołub, Tomasz Czakiert (2013), 'Comparison of the standard Euler–Euler and hybrid Euler–Lagrangeapproaches for modeling particle transport in a pilot-scale circulatingfluidized bed', Czestochowa University of Technology, Czestochowa, Poland, Particuology, Vol 539, pp 1-9.