FMFP14 – K5- 666

COMPUTATIONAL STUDIES ON EFFECTS OF NOVEL GEOMETRIES OF DISTRIBUTOR PLATES ON FLUID DYNAMICS CHARACTERISTICS OF CIRCULATING FLUIDIZED BED RISER

K.P. Shete P. A. Kulkarni and R.S. Patil

In the present paper, CFD simulations using ANSYS-Fluent 14.5 were accomplished to study the effect of distributor plates' geometry on fluid dynamics characteristics like pressure drop along the riser and distributor, suspension density variations along the riser of the Circulating Fluidized Bed (CFB). 3D CFD simulations were performed on the CFB riser of cross section 0.15 m × 0.15 m and height 2.85 m. Modeling and meshing were done using ProE and ANSYS ICEM CFD software, respectively. RNG k-ε model was used for turbulence modeling. Eulerian model with Syamlal O'brien phase interaction scheme was used to simulate the two phase flow (air
+ sand mixture flow). Modeling and simulation were performed for perforated distributor plate and results obtained were compared with available experimental data. In this way, after validation of computational results, further CFD simulations were performed for novel geometry of distributor plate. Results obtained for different distributor plates were compared for pressure drop and suspension density variation.

Computational Studies on Effects of Novel Geometries of Distributor Plates on Fluid Dynamics Characteristics of Circulating Fluidized Bed Riser

K.P. Shete[1], P. A. Kulkarni[1], R.S. Patil[1]

[1]Birla Institute of Technolgy and Science Pilani – K K Birla Goa Campus, India
ranjitp@goa.bits-pilani.ac.in

Abstract. In the present paper, CFD simulations using ANSYS-Fluent 14.5 were accomplished to study the effect of distributor plates' geometry on fluid dynamics characteristics like pressure drop along the riser and distributor, suspension density variations along the riser of the Circulating Fluidized Bed (CFB). 3D CFD simulations were performed on the CFB riser of cross section 0.15 m × 0.15 m and height 2.85 m. Modeling and meshing were done using ProE and ANSYS ICEM CFD software, respectively. RNG k-ε model was used for turbulence modeling. Eulerian model with Syamlal O'brien phase interaction scheme was used to simulate the two phase flow (air + sand mixture flow). Modeling and simulation were performed for perforated distributor plate and results obtained were compared with available experimental data. In this way, after validation of computational results, further CFD simulations were performed for novel geometry of distributor plate. Results obtained for different distributor plates were compared for pressure drop and suspension density variation.

Keywords: CFB Riser, Distributor Plates, Two Phase Flow, CFD Simulations

1 Introduction

Present paper in a broad way is associated with energy sector - power plant engineering. In this domain, it is concerned with Circulating Fluidized Bed (CFB). CFB has several applications and can be used as a boiler (heat exchanger), dryer, gasifier, combustor etc. A large number of CFB units are installed for power generation throughout the world [1]. In different application areas of CFB, many researchers reported the heat transfer characteristics are the function of fluid dynamic characteristics [2–10]. Hence it is important to study the fluid dynamic characteristics of a CFB riser.

Large numbers of papers are available on experimental studies to study the effects of distributor plate design on bed hydrodynamics [11-19]. One of the fluid dynamic characteristics such as pressure drop across the distributor plate affects the pressure drop in a CFB riser, hence it is the critical parameter in designing any CFB unit for its different application areas [11-19]. Also for a given operating conditions, particles'

5th International and 41st National Conference on Fluid Mechanics and Fluid Power,
12th December – 14th December 2014, Indian Institute of Technology Kanpur, India

distribution of the material (sand or coal etc) across the riser and along the height of riser varies based upon the density of particles, different designs of distributor plate, and percentage opening area in the distributor plate [11-22]. Also proper design of the distributor may significantly reduce the auxiliary power consumption and affects economics [23]. In addition, most fluidized beds have to deal with the problems of non-uniform fluidization and back-sifting of solids [23].

Hence with above discussion it is clear that distributor plate's design affects hydrodynamics characteristics and subsequently heat transfer characteristics in many application areas of CFB. Therefore a good rational design of the distributor plate is very important from an operational as well as an economic standpoint of CFB. However numerical studies (2D CFD simulations) on effects of distributor plate design on hydrodynamic characteristics are limited to few papers only [20-22]. Therefore present paper aims to study numerically (3D CFD simulations) the effects of existing design (perforated vertical cylindrical pathways) and novel design (inclined cylindrical pathways) of distributor plates on fluid dynamic characteristics of a CFB riser. Also, through literature review, it is observed that dead zones formation on the upper surface of distributor take place in each corner of the riser using the perforated distributor as shown in Fig. 1. Swirl flow will not be generated by the geometry as shown in Fig.1 because all cylinders were kept perpendicular to its base hence horizontal component of velocity causing generation of swirl flow was absent. Hence new design as shown in Fig. 2 has been proposed to avoid formation of the dead zones and hence to increase the suspension density with increase in height of the riser.

2 Modeling and Grid Independence Test

A riser with perforated distributor of cross section 0.15 m × 0.15 m and height 2.85 m was used for simulation purpose because experimental data was available for validation purpose. In the riser geometry as shown in Fig. 3, return leg (mass inlet) is of cross section 0.15 m × 0.15 m with its bottom end located 0.225 m above the distributor plate and the gas outlet is of cross section 0.15 m × 0.15 m with its center located at a distance of 0.075 m from riser top. A perforated distributor plate as shown in Fig. 1 has thickness 5 mm and 456 orifices. Each orifice of 4 mm diameter and pitch 7 mm causes 25.46% opening area in the distributor plate. Novel distributor has 5 mm thickness and total 440 orifices. Each orifice of 4 mm diameter, pitch 6.5 mm, tilted at an angle of 45^0 to form inclined cylindrical pathways. Novel design of a distributor plate has inclined cylindrical pathways, which were divided into four groups. Orientation of incliation of cylinders of different four groups is made in such a way that swirling flow will be obtained in the riser. Novel distributor has a core orifice (cylinder) of radius 8 mm. This cylinder will not contribute to the generation of swirl, however will help in generating central core (cylindrical) air path which helps to maintain swirl flow around it because of its cylindrical shape. Because of swirl flow formation due to horizontal component of velocity, dead zones formation in the riser will be reduced.

The percentage opening area is the same for both distributor plates (25.46%). Meshing was done with ANSYS ICEM CFD. A hybrid mesh (tetrahedral cell near to wall and hexahedral in the center) with 378427 nodes was used for the riser having perforated distributor plate with a cell size of 10 mm in the central region. A hybrid mesh (prismatic near to wall, hexahedral in the center, and tetrahedral cells used to connect prismatic and hexahedral cells) with 404082 nodes was used for the riser having novel distributor plate with a cell size of 10 mm in the central region. Prism layers increases node number by large amount. Prism mesh layer were added at walls of riser to account for higher velocities (larger y+) and swirling effects that are caused due to inclined pathways in the distributor.

ANSYS Fluent 14.5 was used for simulation. 3D CFD unsteady simulations were performed. The continuity and momentum equations for the solid and fluid phase were solved. Pressure based solver was used. RNG k-ε model was used for turbulence modeling of the flow inside the riser [24]. RNG turbulence model has a calculation for effective viscosity. It can also be used for swirling flows and as the aim of this work is to simulate swirling flow due to novel distributors. Y^+ value was maintained below 100 for all simulations. To deal with low Y^+ values, scalable wall function was used [25].Scalable wall function was used to ensure proper application of wall function to areas which have Y^+ values less than 30.Gas-solid two phase flow was represented by Eulerian-Eulerian i.e. two fluid model with air as the primary phase and sand as the secondary granular phase [26]. In this case, the drag force on sand due to air was specified by a correlation. Drag correlation calculates drag coefficient based on slip velocity. This value is then used to calculate forces on the secondary phase. The Syamlal-Obrien drag correlation was used in the simulation [27]. Granular phase (sand) was defined as size of 0.461 mm which was the mean diameter of particles in experimental data [28] with Syamlal-O'Brien granular viscosity model, solids pressure described by Lun-et-al's model. The collision restitution coefficient was set to 0.9. Boundary conditions used were as follows-inlet velocity as 16.24 m/s and pressure 0Pa, inlet turbulence intensity as 5%, gas outlet as pressure-outlet with gauge pressure 0Pa, mass inlet with a mass flow rate of 0.438 kg/s with a volume fraction of 0.6, initial bed with volume fraction as 0.62, static bed height of 19.6 cm above the distributor plate. Phase coupled SIMPLE pressure-velocity coupling was used. Discretization scheme used was first order upwind. Under-relaxation factors used were 0.1 for momentum, 0.5 for turbulent kinetic energy, and 0.6 for turbulent dissipation rate. Hybrid initialization was used with a time step of 0.01 sec for both simulations.

A grid of 254267 and 378427 nodes were used for simulation of riser having perforated distributor at its bottom. As shown in Fig. 4, the pressure drop along the height of the riser i.e along the three sections, each of 60 cm with midpoints at 0.9 m, 1.5 m, and 2.1 m above the distributor were plotted for the purpose of validation.

Existing regular geometry of normal distributor having cylindrical air pathways of 4 mm diameter perpendicular (90^0) to the base of the plate:

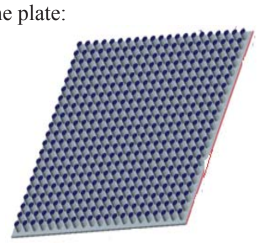

Fig. 1(a). 3D view of normal distributor of 0.15 m × 0.15 m

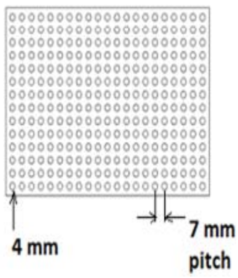

Fig. 1(b). Top view of normal distributor

Novel geometry of distributor (air pathways inclined at 45^0 to base of plate. Four groups of pathways are inclined at 45^0 in different directions to obtain swirl flow):

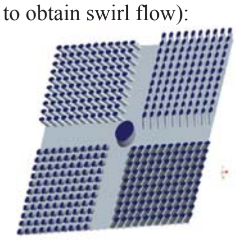

Fig. 2(a). 3D view of novel distributor type 1 & 2 of 0.15 m × 0.15 m

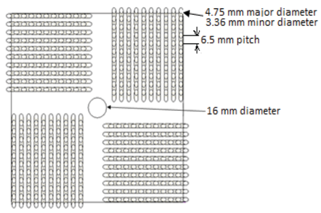

Fig. 2(b). Top view of novel distributor

Fig. 2(c). Side view of novel distributor

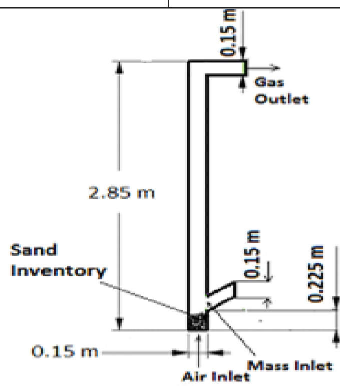

Fig. 3. Riser geometry

The pressure drop for the grid with 254267 nodes deviated from the experimental results [28] in the lower, middle and upper region as shown in Fig. 4. Simulations with 254267 nodes over predict the results. This was because overestimation of drag by the drag model in grids due to larger cells' size in the riser, as reported by [29]. Hence refinement of cells was done for the entire riser, which resulted in smaller cells' size and increases total number of cells and considerable (almost double) computational time of simulations. Total number of nodes increases from 254267 to 378427. Mesh used has hexahedral elements in most of the riser core along with tetrahedral elements near the wall and triangular elements on the wall surfaces. After refinement of the cells to 378427 nodes, results match with published literature both qualitatively and quantitatively than 254267 nodes.

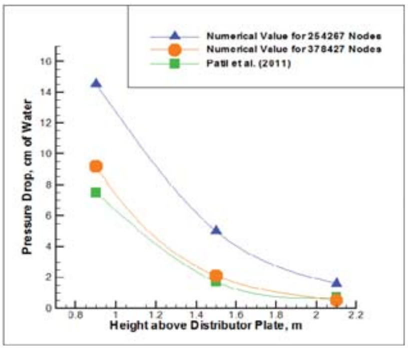

Fig. 4. Pressure drop above distributor plate, perforated distributor

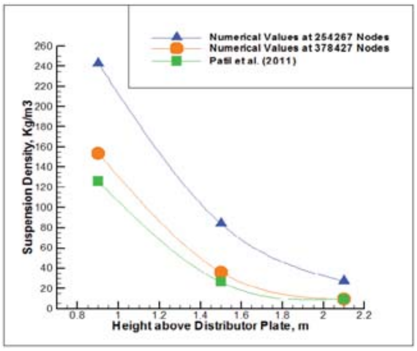

Fig. 5. Suspension density above perforated distributor

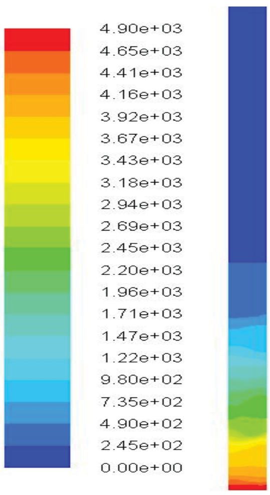

Fig. 6. Static pressure of mixture in Pa

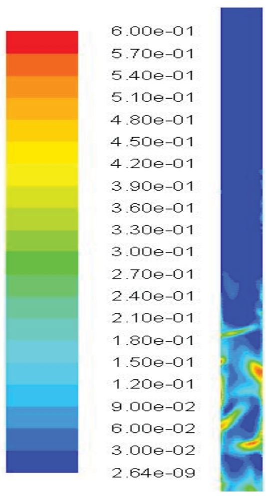

Fig. 7. Volume fraction of sand

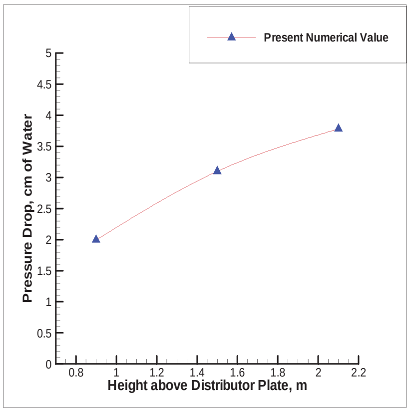

Fig. 8. Pressure drop above distributor plate, novel distributor

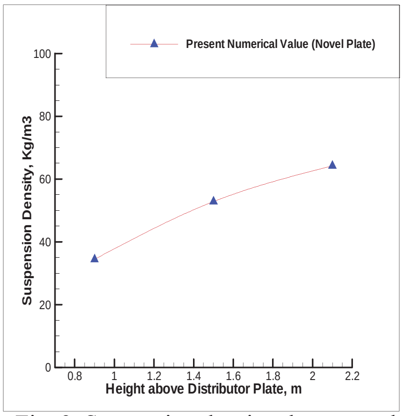

Fig. 9. Suspension density above novel distributor plate

3 Results and Discussion

3.1 Perforated Distributor Plate

Pressure drop along 5 mm thickness of distributor plate was 0.10 cm. As shown in Fig. 5, suspension density shows a similar trend as pressure drop. Suspension density is dependent on pressure drop, and plays a major role in heat transfer between two phase flow (air + sand mixture flow) and water flowing in the riser tubes of boilers. Contours for static pressure and volume fraction of sand were obtained as shown in Fig. 6 and Fig. 7, respectively. Voidage (1-volume fraction of sand) is the only variable used while calculating suspension density.

3.2 Novel Distributor Plate

Pressure drop between Tap 1(0.62 m above distributor plate) and Tap 2 (1.18 m), Tap 3 (1.22 m) and Tap 4 (1.78 m), Tap 5 (1.82 m) and Tap 6 (2.38 m) was plotted corresponding to the lower (0.9 m), middle (1.5 m) and upper region (2.1 m), respectively as shown in Fig. 8. Pressure drop and suspension density show a different trend in the novel distributor plate as compared to the perforated distributor plate as shown in Figs. 4 & 8 and Figs. 5 & 9, respectively. Pressure drop along 5 mm thickness of novel distributor was 3.85 cm, which was around 40 times more than perforated distributor, which clearly indicates that more power will be consumed by blower to lift the sand in the upward direction inside the riser of CFB. The pressure drop values show an increasing trend for the novel distributor plate as shown in Fig. 8 along the height of the riser which is opposite to the trend obtained experimentally and numerically for perforated plate as shown in Fig. 4. Suspension density follows the same pattern as shown in Fig. 9. Reason for having higher suspension density in the upper region of the riser with novel distributor is the swirling effects, the velocity in lower regions is higher and also has its one of the components in horizontal direction, which causes the lower concentration of sand in the lower (bottom) riser section. Also axial velocities decrease to a value of 4 m/s as the riser height increases. Hence, more sand concentration i.e suspension density may be observed in the upper region. The results for novel distributor plate were plotted as shown Fig. 8 and Fig. 9 for a simulation time of 23 sec.

4 Conclusions

Novel distributor plate causes a change in the suspension density trend in fluidized beds having regular perforated plate. The average suspension density in the lower section (0.6 m to 1.2 m above distributor) for perforated plate is 135 kg/m^3 where as in the case of novel distributor; it is 34 kg/m^3. The numerical data predicts suspension density value of 64 kg/m^3 in the upper region of riser (1.8 m to 2.4 m above novel distributor) which was 10 times suspension density value for the perforated distributor. This prediction also suggests that formation of dead zones was reduced with novel distributor plate on the upper surface of distributor near the four corners of riser. It is also observed that pressure drop along 5 mm thickness of novel distributor was around 40 times pressure drop value evaluated for the perforated distributor, which clearly indicates that more power will be consumed by blower to lift the sand in the upward directing inside the riser of CFB.

References

1. M. Hupa.: Current Status and Challenges within Fluidized Bed Combustion. Advanced Combustion and Aero-Thermal Technologies NATO Science for Peace and Security Series C: Environmental Security.vol. 1,pp. 87-101 Springer (2007)
2. P. Basu, P.K. Nag.: An Investigation into Heat Transfer in Circulating Fluidized Beds, Int. J. Heat Mass Transfer. 30 (11),2399-2409 (1987)
3. R.L. Wu, C.J. Lim, J. Chaouki, J.R. Grace: Heat Transfer from a Circulating Fluidized Bed to Membrane Water Wall Cooling Surfaces. AIChE J. 33 (11),1888-1893 (1987)
4. P. Basu, P.K. Nag.: Heat Transfer to Walls of Circulating Fluidized Bed Furnace. Int. J. Heat Mass Transfer 51 (1),1-26 (1996)
5. D. Shi, R. Nicolai, L. Reh.: Wall-to-Bed Heat Transfer in Circulating Fluidized Beds. Chem. Eng. Process. (37),287-293 (1998)
6. W.B. Fox, N.S. Grewal, D.A. Moen.: Wall-to-Bed Heat Transfer in Circulating Fluidized Beds, Int. Commun. Heat Mass Transfer. 26 (4),499-508 (1999)
7. J.D. Pagliuso, G. Lombardi.: Experiments on local heat transfer characteristics of a circulating fluidized bed, Exp. Thermal Fluid Sci. 20,170-179 (2000)
8. A.K. Kolar, R. Sundaresan.: Heat Transfer Characteristics at an Axial Tube in a Circulating Fluidized Bed Riser. Int. J. Thermal Sci. 41,673-681 (2002)

9. R. Sundaresan, A.K. Kolar.: Core Heat Transfer Studies in a Circulating Fluidized Bed. Powder Technol. 124,138-151 (2002)
10. G.R. Grulovic, N.B. Vragolovic, Z. Grbavcic, Z. Arsenijevic.: Wall-to-Bed Heat Transfer in Vertical Hydraulic Transport and in Particulate Fluidized Beds. Int. J. Heat Mass Transfer 51,5942-5948 (2008)
11. D. Sathyamoorthy,C.H. Rao.: Gas Distributors in Fluidized Beds. Powder Technol.20,47-52 (1977)
12. S.C. Saxena, A. Chatterjee, R.C. Patel.: Effect of Distributors on Gas-Solid Fluidization.Powder Technol. 22,191-198 (1979)
13. D. Sathyamoorthy, C.H. Rao.: The Choice of Distributor to Bed Pressure Drop Ratio in Gas Fluidized Beds.Powder Technol.30,139-143 (1981)
14. N. Upadhyay, S.C. Saxena, F.T. Ravello.: Performance Characteristics of Multijet Tuyere Distributor Plate.Powder Technol.30,155-159 (1981)
15. F. Ouyang, O. Levenspiel.: Spiral Distributor for Fluidized Beds. Ind. Chem. Process Des. Dev. 25,504-507 (1983)
16. D. Geldart, J. Baeyens.: The Design of Distributors for Gas-Fluidized Beds. Powder Technol. 42,67-78 (1985)
17. Z. Garncarek, L. Przybylski, J.S.M. Botterill, C.J. Broadbent.: A Quantitative Assessment of the Effect of Distributor Type on Particle Circulation.Powder Technol. 91,209-216 (1997)
18. D. Sathyamoorthy, C.H. Rao.: On the Influence of Aspect Ratio and Distributor in Gas Fluidized Beds.Chemical Engineering Journal 93,151-161 (2003)
19. P. C. Josephkunju.:Influence of Angle of Air Injection and Particles in Bed Hydrodynamics of Swirling Fluidized Bed. PhD Thesis Report, School of Engineering, Cochin University of Science and Technology, Kochi, India.(2008)
20. B. Peng, C. Zhang, J. Zhu.: Numerical Study of the Effect of the Gas and Solids Distributors on the Uniformity of the Radial Solids Concentration Distribution in CFB Risers.Powder Technol. 212, 89-102 (2011)
21. Xi-Zhong Chen, De-Pan Shi, Xi Gao, Zheng-Hong Luo.:A Fundamental CFD Study of the Gas–Solid Flow Field in Fluidized Bed Polymerization Reactors. Powder Technol. 205, 276-288 (2011)
22. B. Peng, J. Xu, J. Zhu, C. Zhang.: Numerical and Experimental Studies on the Flow Multiplicity Phenomenon for Gas–Solids Two-Phase Flows in CFB Risers. Powder Technol. 214, 177-187 (2011)
23. P. Basu.: Combustion and Gasification in Fluidized Bed. Taylor & Francis, LLC (2006)
24. Berker, A., Tulig, T.J.:Hydrodynamics of Gas-Solid Flow in a Catalytic Cracker Riser: Implications for Reactor Selectivity Performance.Chem.Engg. Science 41, 821-827 (1986)
25. Fluent Documentation, ANSYS Fluent 14.5
26. Adnan Almuttahar, FariborzTaghipour.:Computational Fluid Dynamics of High Density Circulating Fluidized Bed riser:Study of modeling parameters. Powder Technol. 185, 11-23 (2008)
27. M. Muthu Kumar, E. Natarajan.:CFD Simulation for Two-Phase Mixing in 2D Fluidized Bed.Int J Adv. Manuf. Technol, (2008)
28. R.S. Patil, M. Pandey, P. Mahanta.:Parametric Studies and Effect of Scale-up on Wall-to-Bed Heat Transfer Characteristics of Circulating Fluidized Bed Risers.Experimental Thermal and Fluid Science 35, 485-494 (2011)
29. Ernst-Ulrich Hartge, Lars Ratschow, Reiner Wischnewski, JoachimWerther.:CFD-Simulation of a Circulating Fluidized Bed Riser. Particuology 7, 283-296 (2009)

FMFP14 – F – 665

EFFECT OF BARREL WALL FIN OF THE CYCLONE SEPARATOR ON FLUID DYNAMIC CHARACTERISTICS

P. A. Kulkarni, K.P. Shete, S. Jogdankar, R.S. Patil

In the present paper, CFD simulations using ANSYS-Fluent 14.5 were accomplished to study the effect of barrel wall fin on cyclone separator's performance in terms of collection efficiency and fluid dynamic characteristics like pressure drop along the height of the cyclone separator, tangential velocity. 3D CFD simulation were accomplished on a Lapple model of cyclone separator with barrel diameter D = 0.2 m with and without fins. Pressure drop along the height of the cyclone separator and collection efficiency for the different particle sizes was evaluated for finned cyclone separator and compared with non-finned cyclone separator. Modeling and meshing were done with ProE and ANSYS ICEM CFD software, respectively. The discrete phase Lagrangian model was used to simulate the gas-solid flow. Computational results were validated using published numerical and experimental data for the cyclone separator without fins, and further CFD simulations were performed for fin based cyclone separator.

Effect of Barrel Wall Fin of the Cyclone Separator on Fluid Dynamic Characteristics

P. A. Kulkarni[1], K.P. Shete[1], S. Jogdankar[2], R.S. Patil[1]

[1]Birla Institute of Technology and Science Pilani – K K Birla Goa Campus, India
ranjitp@goa.bits-pilani.ac.in
[2]Onward Technologies Ltd, Pune

Abstract. In the present paper, CFD simulations using ANSYS-Fluent 14.5 were accomplished to study the effect of barrel wall fin on cyclone separator's performance in terms of collection efficiency and fluid dynamic characteristics like pressure drop along the height of the cyclone separator, tangential velocity. 3D CFD simulation were accomplished on a Lapple model of cyclone separator with barrel diameter D = 0.2 m with and without fins. Pressure drop along the height of the cyclone separator and collection efficiency for the different particle sizes was evaluated for finned cyclone separator and compared with non-finned cyclone separator. Modeling and meshing were done with ProE and ANSYS ICEM CFD software, respectively. The discrete phase Lagrangian model was used to simulate the gas-solid flow. Computational results were validated using published numerical and experimental data for the cyclone separator without fins, and further CFD simulations were performed for fin based cyclone separator.

Keywords: CFB, Cyclone Separator, Fins, Efficiency, CFD Simulations

1 Introduction

The cyclone separator used in the recent technology boiler like Circulating Fluidized Bed (CFB) boiler handles a large volume of gas at high temperature. Therefore the outer skin temperature of the cyclone separator is relatively high; therefore more heat losses occur by natural convection and radiation. To overcome this problem, some cyclone separators are water or steam cooled. This heat recovery can enhance the capacity of the boiler [1]. Therefore proper design of cyclone separator of CFB is essential which subsequently affects fluid dynamic characteristics of cyclone separator. It is observed that heat transfer characteristics of the cyclone separator are the function of fluid dynamic characteristics [2]. Hence it is important to study the fluid dynamic characteristics of a cyclone separator in a CFB system. Literature is also available to predict the fluid dynamic characteristics numerically [3-7].
Major function of the cyclone separator is the separation of the solid particles from gas; in this regard it is important to evaluate the performance of cyclone separator in terms of cyclone efficiency. Literature is available on gas-solid flow structure and

5th International and 41st National Conference on Fluid Mechanics and Fluid Power, 12thDecember – 14th December 2014, Indian Institute of Technology Kanpur, India

collection efficiency of the cyclone separator [8-10]. It is reported that the cyclone efficiency decreases with increase in cyclone separator's diameter, gas outlet duct diameter, gas inlet area [8-10]. A mathematical model for the calculation of cyclone efficiency is reported by Avci and Karagoz [11].

Nag and Gupta [12] reported the effect of fins suspended the cylindrical portion of the cyclone separator on the average heat transfer coefficient for the different operating parameters like solid circulation rate, gas superficial velocity, and pressure in the cyclone separator. Exterior fins have been used on the vortex finder (top gas outlet) of the cyclone separator for enhancing heat transfer [13].

Fins were also used on barrel wall of the cyclone separator for controlling the turbulent conditions [14].

Hence from the above discussion it revealed that lot of literature is available on experimental and numerical studies which predict hydrodynamics and heat transfer characteristics, cyclone efficiency of the cyclone separator without fins [1-11]. Few research papers are available on experimental studies to predict the effect of fins on heat transfer characteristics and turbulent conditions [12-14]. However effect of fins (helical pathway which will carry water or steam) mounted on the inside wall surface of the cylindrical portion of the cyclone separator on collection efficiency and thermal efficiency of CFB boiler has not reported through experimental or numerical study. Arrangement of the fin on the inside wall surface of the cylindrical portion of the cyclone separator will help in heat recovery from the hot gas flowing outside the fin because of possible heat transfer between hot gas and helical fin. However cyclone separator's main function i.e. separating particles from gas should not be affected. Therefore present paper mainly focuses on the numerical study to predict the effect of barrel wall fins on the cyclone efficiency and other fluid dynamic characteristics. Results obtained were compared with cyclone separator without fins.

2 Modelling and Grid Independence Test

The geometry used in this paper was Lapple 2D2D type cyclone separator with the cylindrical section of diameter 200 mm. This cyclone separator is called 2D2D because its barrel height and height of the conical section is twice the diameter of the barrel. Fig. 1 corresponds to the normal cyclone separator. Helical fin with bullet shaped cross-section with a pitch of 40 mm was created on the inside surface of the barrel of the cyclone separator as shown in Fig. 2. Fig. 3 illustrates the geometry of fin. In both the geometries as shown in Fig. 1 and Fig. 2, the height of the cyclone separator is considered along the Y-axis, and the mixture (air + solid) will enter the cyclone separator in a direction along the X-axis. A structured mesh with hexahedral cells was used for the non-finned cyclone separator. This mesh is being created by blocking and O-grid techniques in the ANSYS ICEM CFD software. Due to the highly swirling flow in the cyclone separator, the mesh needs to be finer near the wall for the resolution of near wall effects.

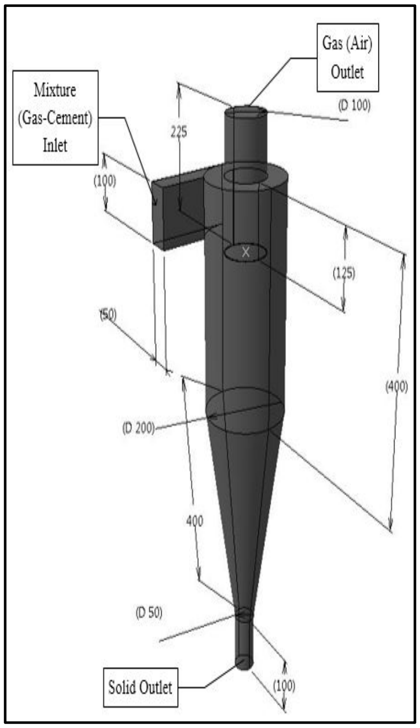

Fig. 1. Geometry of normal (non-finned)cyclone separator (all dimensions in mm)

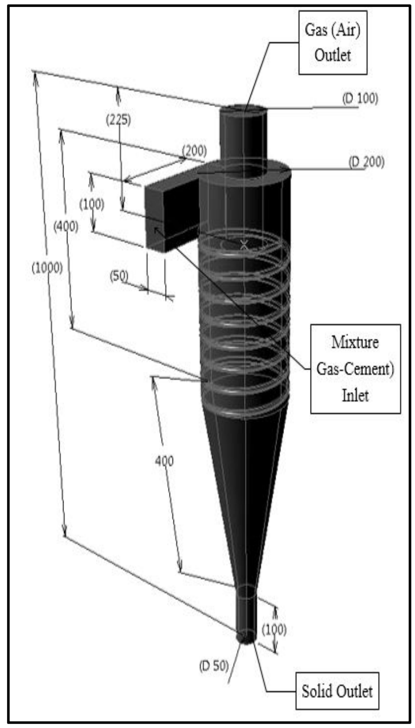

Fig. 2. Geometry of finned cyclone separator (all dimensions in mm)

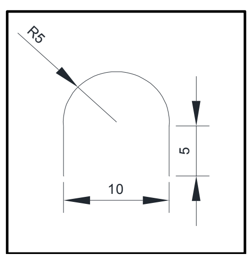

Fig. 3. Cross-section of helical fin (all dimensions in mm)

Three meshes were created for grid independence test, having 490962 nodes (490k), 252276 nodes (252k) and 133468 nodes (133k). The mesh created for the finned cyclone separator is an unstructured hybrid mesh with the core of the cyclone separator having hexahedral cells; the prismatic cells were used near the wall, and the tetrahedral cells were used in between hexahedral and prismatic cells. Mesh density near the wall was higher. 197k, 357k and 571629 nodes (571k) were tested for finned cyclone separator. Nodes 357k and 571k gave very close results, however results obtained using finer mesh i.e. results of 571k nodes are presented in the present paper (Fig. 6 and Fig. 7). Y^+ value measured was around 100 and it is suitable for Reynolds's Stress Model (RSM) model.

The Continuity and the momentum equations were discretized with steady state simulation. The turbulence model used was the Reynolds' Stress Model. The turbulence in the highly swirling flow in the cyclone separator cannot be modelled by using the k-ε models because they were unable to resolve anisotropic stresses and rapidly changing strain rate [3, 4, 6, and 7]. The linear pressure strain sub-model was used for modeling the changing strain rate. For the near wall treatment of the gas flow in the cyclone separator, the scalable wall function was used because it can resolve near wall physics for arbitrarily refined meshes by using the log-law along with the standard wall function. The two phase flow in a cyclone separator is generally modelled using the Eulerian-Lagrangian approach which is implemented in ANSYS Fluent 14.5 as the Discrete Phase Model (DPM). This model was basically chosen because the volume fraction of solid was less than 0.1 and the interaction of solid phase with the gas phase or the continuous phase was not very significant. This model was used for the calculation of the collection efficiency of the cyclone separator. The Discrete Random Walk model was used to count the effect of velocity fluctuations on the forces acting on the particles.

Coupled scheme was used for the pressure velocity coupling. The spatial discretization used was the second order upwind scheme for updating the momentum, turbulent kinetic energy, turbulent dissipation rate, and Reynolds' stresses. Green Gauss node based scheme was used, which calculates the flow field fluxes through the face from average of the variable values at nodes in the mesh instead of the cells.

The pressure outlet boundary condition was given to the exit point where the air exits the cyclone separator. The inlet of the cyclone separator is velocity-inlet with velocity 20 m/s normal to the inlet surface. The solid outlet was the bottom of the cyclone separator that collects the solid particles and had the boundary condition as wall. The cyclone separator wall had a boundary condition as a wall and it reflects the particle from the wall, and the normal and tangential reflection coefficients set at 0.85 and 0.75, respectively. Boundaries of the finned cyclone separator and the normal cyclone separator were same.

The grid independence test was conducted on three different mesh densities of 490962 nodes, 252276 nodes, and 133468 nodes for the normal i.e. non-finned cyclone separator. As shown in Fig. 4 the result shows that the simulation result gives extremely close result for the two higher mesh densities. Hence study had continued using 252k nodes mesh. The simulation results of the tangential velocity magnitude in case of non-finned cyclone separator for the intermediate density (252k nodes) was validated with the experimental data from Wang et al. (2006) and found to be considerably close as shown in Fig. 5. The results were closer to the experimental values

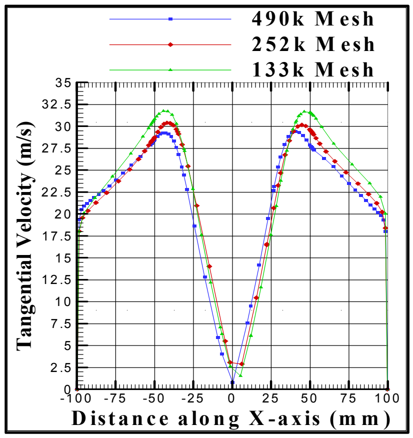

Fig. 4. Grid independence test for normal (non-finned) cyclone separator

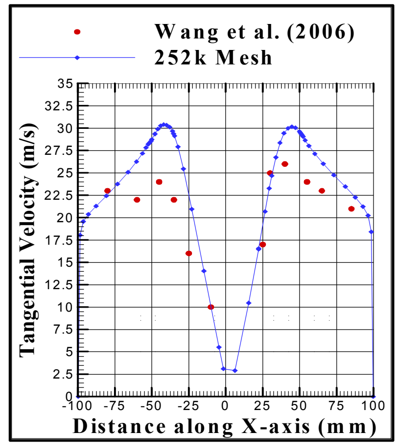

Fig. 5. Validation of tangential velocity using data from Wang et al. (2006)

near the wall while in the central portion; the deviation increases due to the "wall" boundary condition at the solid outlet of the cyclone separator. The tangential velocity in the simulation was taken along the X-axis (parallel to the inlet direction) at a height of 650 mm (Y = 650 mm) from the solid outlet present at the bottom.

3. Results and Discussion

3.1 Pressure Drop along Height and the Pressure Field

The pressure drop along the height (ΔP) as shown in Fig. 6 was obtained by taking difference of total pressure value at the height of 100 mm and 300 mm, 300 mm and 500 mm, 500 mm and 700 mm, 700 mm and 900 mm measured from the bottom (solid outlet) of the cyclone separator, at the negative x extreme along the x-axis in that plane. The total pressure (Fig. 8 and Fig. 9) in the cyclone separator increases from the centre to its wall. The total pressure at the centre of the cyclone separator is lower than the atmospheric pressure because the highly swirling flow creates a vortex around the central region. The pressure drop (Fig. 6) for the fin-based cyclone separator has increased slightly in the cyclindrical region in comparison to the pressure drop in the non-finned cyclone separator. This is because of the helical fin that is located on the barrel wall. The fin creates an obstruction to the flow of gas (air) leading to higher dissipation of dynamic pressure. This can be explained by the increase in turbulence observed in the turbulence intensity contours as shown in Fig. 10 and Fig. 11.

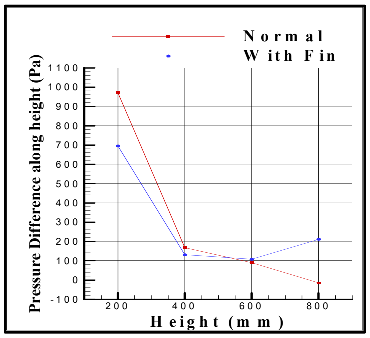

Fig. 6. Pressure drop between discrete points on cyclone wall along the height

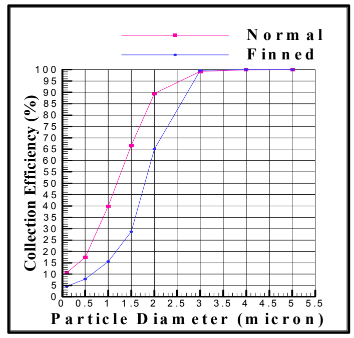

Fig. 7. Cyclone efficiency for normal (non-finned) and finned cyclone separator

The increase in the turbulence intensity in the barrel region also supports that the boundary layer has become more turbulent for the finned cyclone separator due to the addition of the helical fin. This leads to a higher pressure drop in the upper region of the cyclone separator.

3.2 Collection Efficiency

The collection efficiency of the normal and finned cyclone separator was calculated using the discrete phase model (DPM) with injections of 15000 particles of cement (ρ = 3320 kg/m^3) sent as in [7], but the diameters used were 0.1 µm, 0.5 µm, 1 µm, 2 µm, 3 µm, 4 µm, 5 µm, 10 µm. It is observed that collection efficiency increases with increase in particle size as shown in Fig. 7, because larger size particles were heavier hence collected at solid bottom outlet of cyclone separator. As shown in Fig. 7, it is observed that for the same particle size, cyclone efficiency decreases from the normal (non-finned) cyclone separator to the finned cyclone separator. This is because as explained in previous section 3.1 the higher pressure drop in the upper region (cylindrical region) of the finned cyclone separator causes lower intensity of swirling flow in its outer vortex, thus causing a reduction in the tangential velocity of the gas (air). Also the helical fin directly affects the axial velocity, which reduces in the downward direction in the outer vortex near the wall where particles concentration was high. Due to the reduction in axial and tangential velocity, possibility of particles' trapping in the inner vortex of air moving in upward direction increases. Also addition of fins causing a change in the effective diameter of the finned cyclone separator which subsequently causes the reversal of the air at a higher height (refer Figs. 8-9) above the solid outlet in the finned cyclone separator than the normal non-finned cyclone separator causing more particles to escape through the gas-outlet.

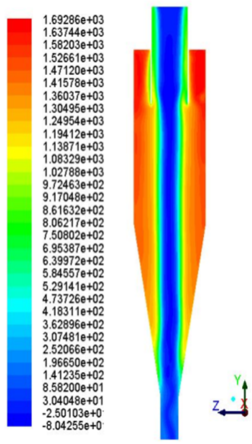

Fig. 8. Total pressure (Pa) contour for cyclone separator without fins

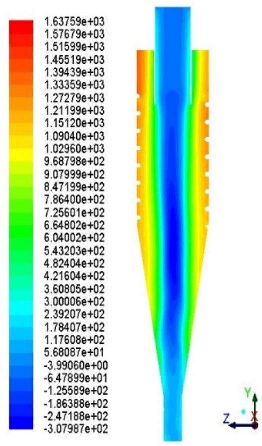

Fig. 9. Total pressure (Pa) contour for finned cyclone separator

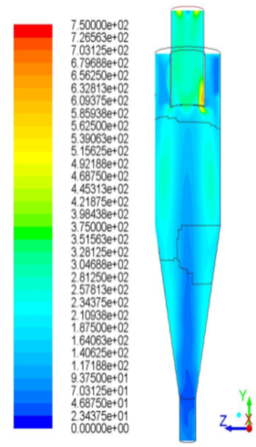

Fig. 10. Turbulent intensity (%) contour for cyclone separator without fins

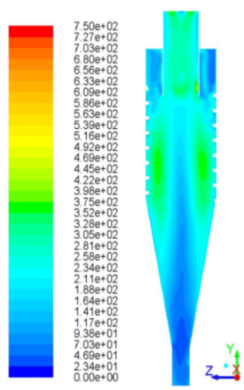

Fig. 11. Turbulent intensity (%) contour for cyclone separator with fins

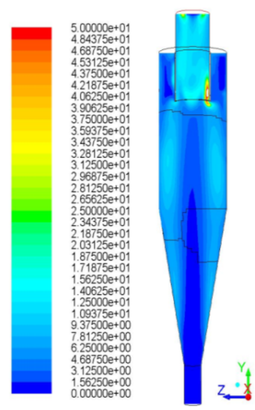

Fig. 12. Turbulent kinetic energy (m^2/s^2) contour for cyclone separator without fins

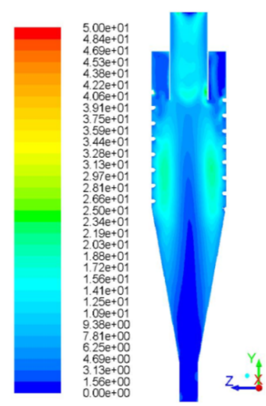

Fig. 13. Turbulent kinetic energy (m^2/s^2) contour for cyclone separator with fins

4. Conclusions

It is observed that the helical fin attached to the barrel wall changes the hydrodynamics of the mixture (air + solid) flow in the cyclone separator significantly. The velocity field was drastically modified due to the helical fin as it is observed that increase in the axial velocity in the upward direction inside the central vortex in comparison with normal non-finned cyclone separator. The helical fin also increases the pressure drop along the height of the cylindrical portion of the finned cyclone separator which subsequently decreases the intensity of swirling in the outer vortex hence tangential velocity decreases. Because of these reasons the particles entrained in the gas (air) upward inner vortex flow and subsequently the cyclone efficiency shows a marginal decrease when the fins are attached to the barrel wall. Hence the pitch (40 mm) used for the present fins should be considered extremely critical because the interaction of the fin with the continuous phase (air) affects the hydrodynamics and hence the cyclone separator's performance (collection efficiency). Therefore changes in the value of the pitch of the fins should be made in such a way that the cyclone efficiency of the finned cyclone separator will increase and number of turns taken by the gas flow in its cylindrical portion may decrease. However for any value of pitch, the increased surface area because of added fins (helical pathway carrying water or steam) on the inside wall of barrel allows a possibility of heat recovery as heat transfer will occur between outside hot gas and fluid flowing inside fin (helical pathway) and hence will enhance the capacity of the CFB boiler.

References

1. Gupta, A.V.S.S.K.S. and P.K. Nag.: Prediction of heat transfer coefficient in the cyclone separator of a CFB. International Journal of Energy Research 24: 1065-1079 (2000)
2. Nag, P.K. and N.K. Singh.: Heat transfer in the cyclone separator of a circulating fluidized bed. In Preprints of CFB-5, 5th International Conference on Circulating Fluidized Beds. Beijing, P.R. China, HM II. (1996)
3. Azadi, M., Azadi, M., & Mohebbi, A.: A CFD study of the effect of cyclone size on its performance parameters. Journal of Hazardous Materials, 182(1-3), 835–41 (2010)
4. Hoekstra, A. J., Derksen, J. J., & Akker, H. E. A. Van Den. (1999). An experimental and numerical study of turbulent swirling flow in gas cyclones, 54, 2055–2065.
5. Shi, L., &Bayless, D. J.: Comparison of boundary conditions for predicting the collection efficiency of cyclones. Powder Technology, 173(1), 29–37 (2007)
6. Wan, G., Sun, G., Xue, X., & Shi, M.: Solids concentration simulation of different size particles in a cyclone separator. Powder Technology, 183(1), 94–104 (2008)
7. Wang, B., Xu, D. L., Chu, K. W., & Yu, a. B.: Numerical study of gas–solid flow in a cyclone separator. Applied Mathematical Modelling, 30(11), 1326–1342. (2006)
8. Dietz, P.W.: Collection efficiency of cyclone separator. AIchE Journal 27: 882-892 (1981)
9. Trefz, M. and E. Muschelknautz.: Extended cyclone theory for gas flows with high solid concentrations. Chemical Engineering Technology 16: 153-160 (1993)
10. Zhou, L.X. and Soo, S.L.: Gas-solid flow and collection of solids in a c clone separator. Powder Technology 63: 45-53 (1990)
11. Avci, A. and I. Karagoz.: Effects of flow and geometrical parameters on the collection efficiency in cyclone separators. Journal of Aerosol Science 34: 937-955 (2003)
12. Nag, P.K. and Gupta, A.V.S.S.K.S.: Fin heat transfer studies in a cyclone separator of a circulating fluidized bed. Heat Transfer Engineering 20: 28-34 (1999)
13. Bodo, K.: Cyclone Separator Vortex Finder with exterior Heat fins. Canadian Patent File, Patent File Number: 780525. Application Number:304031 (1977)
14. Otto, L., Oskar, W.: Flow Control. Master's Thesis Luleå University of Technology, Sweden (2013)
15. ANSYS Fluent 14.5 Documentation. ANSYS Inc. (2012).