MODELLING OF BAFFLES IN ELECTROSTATIC PRECIPITATOR (ESP) TO ACHIEVE OPTIMUM FLOW DISTRIBUTION

Sayem A. S. M. [1*], Khan, M.M.K.[1], M.G. Rasul[1], N.M.S. Hassan[1] and M.T.O. Amanullah[2]
1*School of Engineering and Technology, Central Queensland University, QLD 4702, Australia
[2]Deakin University, Melbourne, VIC 3220, Australia
Email: a.sayem@cqu.edu.au

ABSTRACT

Electrostatic Precipitators (ESP) are the most reliable emission control devices that are used in coal fired power plants to capture fine particles for reducing exhaust emission. Its efficiency is more than 99% or more. However, capturing submicron particles are still a problem due to complex flow distributions and design limitations of ESP. In this study, two different shapes of baffles inside the ESP have been considered to assess their influence on the flow pattern using computational fluid dynamics (CFD) code 'ANSYS FLUENT'.. Due to different shapes, the flow distribution will be changed inside the ESP which is expected to affect and increase the residence time of flue gas. The results of this paper indicate that the proposed shapes can influence in collecting more fine particles

INTRODUCTION

Most of the coal power plants and other process industries generally use Electrostatic Precipitators (ESP) because of their effectiveness and reliability in controlling emission of particulate matter. Before being released into the environment, flue gas flows through the ESP where particles are collected. The ESP is used as a cleaning device. For separating the dust particles from the flue gas, an electrical field is created and used by the ESP. A rectangular collection chamber which is known as the inlet evase and an outlet convergent duct known as the outlet evase are the key components of an ESP. For flow distribution, perforated plates are placed inside the inlet and outlet evase. A number of discharge electrodes (DE) and collection electrodes (CE) are positioned inside the collection chamber. Figure 1 presents an ESP arrangement and shows the section of a typical wire-plate ESP channel where a set of discharge electrodes is suspended vertically and the gas flows through this channel. By using an electric field, particles from the flow are attracted to the electrodes and thus they are separated. In this paper the influence of different shapes baffles on flow pattern is discussed. Flow pattern has a significant impact on particle collection and is also an important parameter for designing and adjusting the operation of an ESP [1].

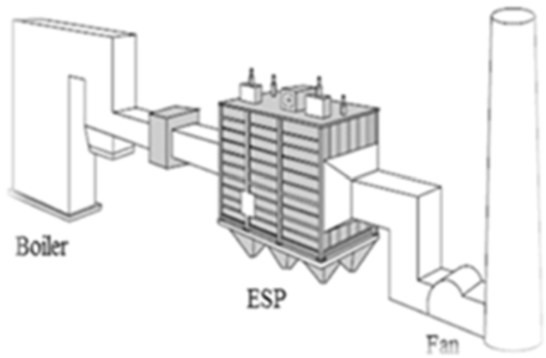

Figure 1 A typical arrangement of an ESP in the power plant[1]

Particle emissions have become one of the major concerns to power industry [2]. Particulates contain such materials (PM > 2.5) that can affect our health severely because they may go into the deeper parts of the respiratory tract [3].

Power stations always desire to control the particulate emissions at a minimum cost in spite of having 99.5% capture capability of particles bigger than PM>2.5 micron by the electrostatic precipitators. Currently, the particles of size PM (Particulate Matter) 2.5 or less may escape the ESP. However, it is anticipated that the new EPA regulation will soon be imposed for mandatory capture of these particles.

Recent studies that have been undertaken on ESP modelling include work by Shah [4] Gallimberti [5], C. Ruttanachot [6], B.Y. Guo [7], Z. Al-Hamouz[8], Sayem [9]. It appears that there is a need to further improve ESP's ability and efficiency by capturing these smaller particles. One idea of achieving this is to increase the particle residence time within the ESP. Introduction of column of baffles to increase the residence time of exhaust fluid flow inside the ESP has been considered in this study. Fluid flow which is influenced by vorticity created by the baffles has been examined by a computational fluid dynamics (CFD) analysis.

GEOMETRY

A laboratory scale ESP model, geometrically similar to an industrial ESP, was designed and fabricated by Shah et al [4, 10] at the Thermodynamics Laboratory of CQUniversity, Australia to examine the flow behaviour inside the ESP. This laboratory scale ESP consisted of a rectangular collection chamber and an inlet evase and an outlet evase. The current study focuses on further improvement of the flow behaviour inside the ESP and has taken into account the rectangular collection chamber as a rectangular duct including inlet evase, duct and outlet evase, outlet duct for simplicity of the model geometry. This is because the dust particle separation from the flue gas and the collection of dust particles usually occur inside the rectangular duct. In this paper, the flow behaviour by changing the baffle shape has been investigated. Two different types of baffles in ESP duct have been considered. First set of baffles are of arrow shape and second sets are of circular shape .The Geometry is drawn in Design Modeller of ANSYS Fluent and further processing is done for meshing and refinement The geometry is shown in Fig. 3(a) and Fig. 3(b). ANSYS code 'FLUENT' is used for numerical simulation of fluid flow behaviour.

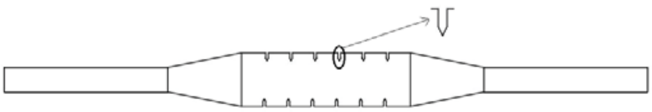

Figure 3(a) ESP containing arrow shape baffles

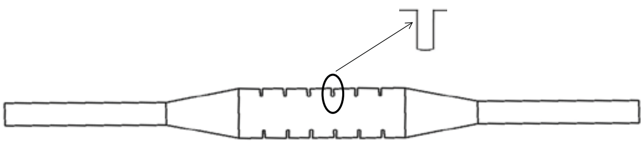

Figure 3(b) ESP containing circular shape baffles

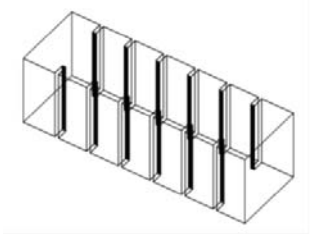

Figure 3(c) 3d view of ESP duct consists of circular shape baffles

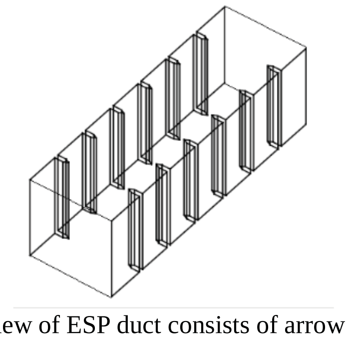

Figure 3(d) 3d view of ESP duct consists of arrow shape baffles

The ESP considered in this paper has a single chamber. Six circular and arrow shape baffles are inserted in two opposite sidewalls of two separate ducts. Fig. 3(c) and Fig. 3(d) represent circular baffles and arrow shape baffles respectively. The length, width and height of the duct are: 157.5 cm, 50 cm and 50 cm respectively. The baffles are equally spaced and the distance between two baffles is 20 cm. The dimension of each baffle is: height 50 cm, width 5 cm and length 7.5 cm. (pick point)

It is noted that the results of the impact of the baffles on the flow are discussed on a qualitative basis since more results are required for a quantitative analysis.

NUMERICAL APPROACH AND SIMULATION PROCEDURE

As mentioned earlier, the mesh created by Design Modeller is exported to ANSYS to discretize the fluid domain into small cells to form a volume mesh or grid and set up appropriate boundary conditions. Numerical computation of fluid transport includes continuity, momentum and turbulence model equations. The flow properties and equations are solved and analyzed by CFD code "FLUENT"[11]

Continuity equation:

$$\frac{\partial \bar{u}}{\partial x}+\frac{\partial \bar{v}}{\partial y}+\frac{\partial \bar{w}}{\partial z}=\frac{\partial (u_i)}{\partial x_i}=0 \tag{1}$$

Momentum Equation:

$$\frac{\partial}{\partial t}(\rho u_i)+\frac{\partial}{\partial x_j}(\rho u_i u_j)=-\frac{\partial p}{\partial x_j}+\frac{\partial \tau_{ij}}{\partial x_j}+\rho g_i+F_i \tag{2}$$

In this equation, p is static pressure and self-defined source term are contained in Fi, Stress tensor is determined by the following equation:

$$\tau_{ij}=\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial v_j}{\partial x_i}\right)\right]-\frac{2}{3}\mu\frac{\partial u_i}{\partial x_i}\delta_{ij} \tag{3}$$

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_j}(\rho k u_j)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu t}{\sigma k}\right)\frac{\partial k}{\partial x_j}\right]+(G_k+G_B-Y_M)-\rho\epsilon+S_k \tag{4}$$

$$\frac{\partial}{\partial t}(\rho\epsilon)+\frac{\partial}{\partial x_j}(\rho\epsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{u_t}{\sigma\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right]+C_{1\epsilon}\frac{\epsilon}{k}(G_k+C_{3\epsilon}G_b)-C_{2\epsilon}\rho\frac{\epsilon^2}{k}+S_\epsilon \tag{5}$$

Where u, v, w is each component of gas velocity, (m/s) and μ, μ_t are molecular viscosity and kinetic viscosity (Pa.s) respectively; ρ (kg/m^3) is fluid density, G_k represents turbulent energy generated by mean velocity gradient; G_B is turbulent energy generated by buoyancy; Y_M represents pulsation expansion in turbulent model of compressible flow; $C_{1\epsilon}$, $C_{2\epsilon}$ and $C_{3\epsilon}$ are empirical constant; σk and $\sigma\epsilon$ are corresponding turbulent Prandtl number in k-equation and ε-equation; S_k and S_ε are self-defining source term; $C_{1\epsilon}$ =1.44, $C_{2\epsilon}$ =1.92, $C_{3\epsilon}$ =0.09, σk =1.0, $\sigma\epsilon$ =1.3[12]

Realizable - model was selected in this article, as it differs from the others - model in two important ways. Firstly, the realizable - model contains a new formulation for the turbulent viscosity. Secondly, a new transport equation for the dissipation rate, ε (dissipation rate of turbulent kinetic energy, m^2 /s^3), has been derived from an exact equation for the transport of the mean-square velocity fluctuation [10]. The turbulent kinetic energy (turbulent kinetic energy, m^2 /s^2) and its dissipation rate for Realizable - model are obtained from the transport equation (4) and equation (5) respectively [11, 12].

MESH GENERATION AND BOUNDARY CONDITION

Three dimensional simplified model of ESP was employed in this study to investigate the flow properties. ANSYS 15.0 was used to establish models for calculating regions where unstructured grids were generated and finally, after resizing and refinement, structured grid was obtained. The number of nodes and cells obtained for circular shape baffles were 28,277, 47,952 and for arrow shape baffles were 27,258, 46,388 respectively. Air was used as the fluid and its acquiescent properties were maintained with a constant velocity. The boundary conditions were applied as follows: inlet velocity was considered as constant at 10 m/s, the outlet boundary condition was pressure outlet and "No slip" (velocity at the wall to be zero) boundary condition was imposed on the side walls including baffle's side face and front face.

The finite volume methods were used to discretize the partial differential equations of the model. The Semi-Implicit Method for Pressure - Velocity Coupling Equations (COUPLED) scheme was used for pressure–velocity coupling and the first order upwind scheme was used because of its combination of accuracy and stability and as this scheme interpolates the variables on the surface of the control volume. Turbulent kinetic energy k and turbulent dissipation rate ε were considered as a first order upwind for better simulation accuracy.

Figure 4. (a) Structured mesh for arrow shape baffles (2D)

Figure 4. (b) Structured mesh for circular shape baffles (2D)

In order to compute the results, all simulations were carried out on an Intel Core i5 processor computer that has 2.80 GHz processor and 8.00 GB of RAM, 64-bit operating system.

RESULTS AND ANALYSIS

The assembled graphs below are the simulation results of velocity distribution and pressure distribution for circular shape and arrow shape respectively. Six baffles were inserted into each wall in air flow distribution plates. In the following figures, the influence of baffle on forming skewed air flow pattern is considered only.

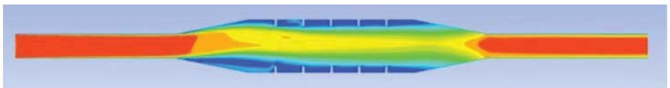

Figure 5(a) Velocity distributions for circular shape baffles

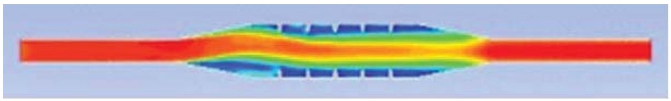

Figure 5 (b) Velocity distributions for arrow shape baffles

Figure 5 (c) Pressure distribution for circular shape baffles

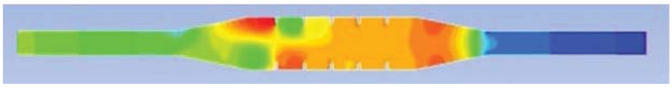

Figure 5 (d) Pressure distributions for arrow shape baffles

Fig. 5(a) and 5(b) show the contour of the velocity distribution for circular shape and arrow shape baffles in ESP respectively. It is seen from these figures that the velocity distribution pattern is quite different because of the changing shapes of baffles. In case of circular shape baffles, the velocity stream line is changed more frequently compared to the arrow shape -baffles velocity distribution even though the arrow shape baffles velocity distribution maintains a smooth stream line.. From the view point of velocity pattern, circular shape baffles have greater potential to capture particulate matter since the velocity changes more frequently causing pressure fluctuation and hence dropping of the particles from flow. From figure 5(a) it is found that the area near the baffles has almost zero velocity, which enables to collect more finer particles and to increase residence time as well. However, it increases drag force for upcoming flue gas. Arrow shape baffles velocity pattern keeps a smooth centerline velocity stream which reduces drag force for upcoming flue gas and also decreases residence time due to smooth velocity pattern shown in 5 (b). Figure 5(c) and 5(d) show the pressure distribution for circular shape and arrow shape baffles respectively. In case of circular shape, pressure increases gradually and reaches a maximize at the middle of ESP where as in arrow shape baffles a mixed pressure distribution is obtained. From these figures it is also observed that near the back face of the baffles, the velocity is small or near zero. This indicates that more dust is likely to settle and accumulate there.

CONCLUSION AND RECOMMENDATION

The CFD analysis of two different shapes baffles ESP shows some encouraging and favorable result. The summery that can be drawn is as below:

• Qualitative analysis of two different flow pattern shows that installing baffles forms skewed gas flow whose formation is influenced by baffles shape, interval between two baffles. Skewed gas flow also increases the residence time of the flue gas inside the duct leading to more dust collection. Further study will be conducted on the gap between two opposite baffles or placing the baffles on an incline inside the duct.

• Velocity distribution indicates that smooth velocity pattern is obtained in case of arrow shape baffles whereas circular shape baffles offer comparatively lower velocity adjacent to the baffles, which is more suitable for dust collection. Further study is needed to examine the residence time and drag force for these baffles.

• Arrow shape baffles offer lower drag force whereas circular shape offers more residence time. Therefore, it is needed to optimize both the models to get the desired flow and residence time to enable dust collection.

• The dynamics of formation of the skewed flow is complicated and is not quantitatively analyzed. However, qualitatively, it shows the creation of vortex flow near the baffles and existence of a near zero velocity close to the wall of the baffles, which suggests an improvement of the dust collection efficiency.

• Further investigation is required to determine the strength of vortex flow as well as l time for dust collection from the wall. This is because, as the thickness of dust collection is increased, further interaction with the spinning motion of flow may wipe out the accumulated dust in the wall.

REFERENCES

[1] M. HU, X. SUN, C. MA, Y. LIU, L.-q. WANG, Numerical Simulation of Influence of Baffler in Electric Field Entrance to Form Skewed Gas Flow.

[2] APTI, Air Pollution Training Institute (APTI) Course SI: 412B. 1998., in, U.S Environmental Protection Agency., 1998.

[3] L. Morawska, V. Agranovski, Z. Ristovski, M. Jamriska, Effect of face velocity and the nature of aerosol on the collection of submicrometer particles by electrostatic precipitator, Indoor air, 12 (2002) 129-137.

[4] S.M.E. Haque, M.G. Rasul, A.V. Deev, M.M.K. Khan, N. Subaschandar, Flow simulation in an electrostatic precipitator of a thermal power plant, Applied Thermal Engineering, 29 (2009) 2037-2042.

[5] J. Podlinski, A. Niewulis, J. Mizeraczyk, P. Atten, ESP performance for various dust densities, Journal of Electrostatics, 66 (2008) 246-253.

[6] C. Ruttanachot, Y. Tirawanichakul, P. Tekasakul, Application of electrostatic precipitator in collection of smoke aerosol particles from wood combustion, Aerosol and Air Quality Resarch, 11 (2011) 90-98.

[7] B.Y. Guo, Q.F. Hou, A.B. Yu, L.F. Li, J. Guo, Numerical modelling of the gas flow through perforated plates, Chemical Engineering Research and Design, 91 (2013) 403-408.

[8] Z. Al-Hamouz, Numerical and experimental evaluation of fly ash collection efficiency in electrostatic precipitators, Energy Conversion and Management, 79 (2014) 487-497.

[9] A.S.M. Sayem, M.M.K. Khan, M.G. Rasul, M.T.O. Amanullah, N.M.S. Hassan, Effects of baffles on flow distribution in an electrostatic precipitator (ESP) of a coal based power plant, in: 6th BSME International Conference on Thermal Engineering (ICTE 2014), Procedia Engineering, Dhaka, Bangladesh, 2015, pp. 8.

[10] S.M. Haque, M. Rasul, M.M.K. Khan, A. Deev, N. Subaschandar, Influence of the inlet velocity profiles on the prediction of velocity distribution inside an electrostatic precipitator, Experimental Thermal and Fluid Science, 33 (2009) 322-328.

[11] ANSYS FLUENT 12.0 Theory Guide.

[12] F. Dubois, W. Huamo, New advances in computational fluid dynamics—theory, methods and applications [M], in, Beijing: Higher Education Press, 2001.