Effects of baffles on flow distribution in an electrostatic precipitator (ESP) of a coal based power plant

A.S.M. Sayema, M.M.K. Khanb, M.G. Rasula, M.T.O. Amanullahb, N.M.S. Hassanb

aCQU University, Rockhampton, Qld-4702, Australia
bDeakin University, Melbourne, Qld-4702, Australia

Abstract

Electrostatic Precipitators (ESP) are the most reliable and industrially used control devices to capture fine particles for reducing exhaust emission. Its efficiency is 99% or more. However, capturing submicron particles which are hazardous is still a problem as it involves complex flow phenomena and ESP design limitations. In this study, the effect of baffles on flow distribution inside the ESP is investigated computationally. Baffles are expected to increase the residence time of flue gas which helps to collect more particles into the collector plates, and hence increase the collection efficiency of an ESP. Besides, the placement of a baffle is likely to cause swirling of flue gas and hence sub-micron particles move towards the collector plate due to eccentric and electrostatic force. Therefore, the effects of position, shape and thickness of the baffles on collection efficiency which are also important for ESP design are reported in this study. The fluid flow distribution has been modelled using computational fluid dynamics (CFD) software Fluent and the result and outcome are presented and discussed. The result shows that baffles have significant influence on fluid flow pattern and the efficiency of ESP.

1. Introduction

Coal has been a major source of affordable energies for Queensland’s electricity generation for decades. In 2010-11, coal based power plant generated around 76 percent of electricity in all over Australia and 62 percent of it was...
supplied just only to Queensland state and remaining power was supplied to other states [1] which is presented in Fig. 1. Most of the coal power plants and other process industries generally use Electrostatic Precipitators (ESP) because of their effectiveness and reliability in controlling particulate matters. Before going into the environment, flue gas flows through the ESP where dust particles are captured. The ESP can be used as a cleaning device. For separating the dust particles from the flue gas, an electrical force is generally used by the ESP. A rectangular collection chamber which is known as inlet evase and an outlet convergent duct known as 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. Fig. 2 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, particle separation is achieved. In this paper the influence of 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 [2].

Fig. 1. Queensland grid electricity generations by fuel type, 2010-11 (Source: Electricity gas Australia, 2012)

Particle emissions have become one of the major concerns to power industry because of strict rules and regulations of Environmental Protection Agency (EPA). Particulates contain such materials that can affect our health severely as they may go into the deeper parts of the respiratory tract [3]. Performance optimization of the emission control devices replaces energy recovery and conservation methods. Power stations always desire to control the particulate emissions at a minimum cost in spite of having 99.5% capture capability by the electrostatic
precipitators. Currently, the particles of size particulate matter (PM) 2.5 or less may escape through the ESP. However, it is anticipated that new EPA regulation will soon be imposed for mandatory capture of these particles.

Much work has already been done on the gas flow regime within an ESP in recent years [4]. An overview is described by Gallimberti et al. [5]. 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.

2. Geometry

A laboratory scale ESP model, geometrically similar to an industrial ESP, was designed and fabricated by Shah et al. [4, 6] 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 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. The Geometry is drawn in AutoCad 2014 and exported into Fluent for further meshing and refinement. The geometry with dimension is shown in Fig. 3. Rectangular strip is inserted on the wall of the rectangular duct and ANSYS code ‘FLUENT’ is used for numerical simulation of fluid flow behaviour.

The ESP considered in this paper has a single chamber. As seen from Fig. 3(a), six rectangular baffles were inserted in two opposite sidewalls. The rectangular baffles and the side wall are shown in Fig. 3(b) and Fig. 3(c) respectively. The length, width and height of the duct are: 157.5 cm, 60 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 2.5 cm.

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.

3. 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”[7]

Continuity equation:

\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = \frac{\partial (\rho u)}{\partial x} = 0
\]  

(1)

Momentum Equation:

\[
\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i + F_i
\]  

(2)

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

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

(3)

\[
\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu + C_{\mu} \frac{\mu}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + (G_k + G_B - Y_M) - \rho \varepsilon + S_k
\]  

(4)

\[
\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_j}(\rho \varepsilon u_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu + C_{\mu} \frac{\mu}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \varepsilon \left( G_k + C_3 \varepsilon G_B \right) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon
\]  

(5)

where \( u, v, w \) is each component of gas velocity, (m/s) and \( \mu, \mu_t \) are molecular viscosity and kinetic viscosity (Pa.s) respectively; \( \rho \) (kg/m³) 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; \( C_{1\varepsilon}, C_{2\varepsilon}, C_3, \sigma_k, \sigma_\varepsilon \) are empirical constant; \( \sigma_k \) and \( \sigma_\varepsilon \) are corresponding turbulent Prandtl number in \( k \)-equation and \( \varepsilon \)-equation; \( S_k \) and \( S_\varepsilon \) are self-defining source term; \( C_{1\varepsilon} = 1.44, C_{2\varepsilon} = 1.92, C_3 = 0.09, \sigma_k = 1.0, \sigma_\varepsilon = 1.3 \) [8]

Realizable \( k-\varepsilon \) model was selected in this article, as it differs from the others \( k-\varepsilon \) model in two important ways. Firstly, the realizable \( k-\varepsilon \) model contains a new formulation for the turbulent viscosity. Secondly, a new transport equation for the dissipation rate, \( \varepsilon \) (dissipation rate of turbulent kinetic energy, m²/s³), has been derived from an exact equation for the transport of the mean-square velocity fluctuation [7]. The turbulent kinetic energy \( k \) (turbulent kinetic energy, m²/s²) and its dissipation rate \( \varepsilon \) for Realizable \( k-\varepsilon \) model are obtained from the transport equation (4) and equation (5) respectively [7, 8].

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 were 1817424 and 1759268 respectively. Air was used as a fluid and its acquiescent properties were maintained with constant velocity. The boundary conditions were applied as follows: inlet velocity was considered as constant velocity which was 7 m/s, the outlet boundary condition was pressure outlet and "No slip" boundary condition was imposed on the side walls including baffle’s side face and front face.

The finite volume methods were used to discretise the partial differential equations of the model. The Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) scheme was used for pressure–velocity coupling and the second 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 \( \varepsilon \) were considered as a second order upwind for better simulation accuracy.
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.

4. Results and Analysis

The assembled graphs below are the simulation results of velocity distribution and pressure distribution 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.

Fig. 5(a) and 5(b) show the contour of the velocity distribution. It is seen from these figures that the velocity keeps increasing towards the center of the channel between the first two baffles with the reduction of flow area. The velocity however gradually decreases near the wall between the two baffles. An irregular skewed flow pattern is formed at the inlet of the first three baffles, however such flow pattern does not continue towards the outlet of the duct. The skewed motion of flue gas is likely to increase the retention time of flue gas inside the duct. From these figures it is 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.
Formation of skewed flow of flue gas in terms of vectors quantity in 3D view is shown in Fig. 5. (b) Fig. 5. (c). As seen the flow propagation is influenced by the skewed flow in the back and gradually the flow becomes steady.
Fig. 5(d) characterizes the contours of static pressure which shows the creation of a vacuum pressure between first two baffles and Fig. 5 (e) presents the contours of the velocity in the X direction.

Fig. 5 (f) and Fig. 5(g) represent the contours of inlet velocity and outlet velocity distribution respectively which shows that the velocity at the wall is zero and almost uniform at the inlet. Due to the influence of baffles, the velocity profile at the outlet has changed from that at the inlet which is shown in Fig. 5(g). The blue color zone indicates that the velocity remains zero and small from the side wall up to a certain distance. The zero and small velocity zone on both side walls is suitable for more particle sediment on the side wall.

5. Conclusion and recommendation

The preliminary CFD analysis of this investigation shows some encouraging and favorable results. The conclusions that can be drawn are as below:

- Qualitative analysis of the flow pattern shows that installing baffles makes skewed gas flow whose formation is related to the baffle interval and which also increases the residence time of the flue gas inside the duct leading to more dust collection. Further study will be conducted on changing the baffles angle i.e by inserting the baffles or placing the baffles on an incline inside the duct.
The dynamics of formation of the skewed flow is complicated and is not quantitatively analysed. However, qualitatively, it shows creation of vortex flow near the baffles and existence of a near zero velocity close to the wall of the baffles, which assists to improve the dust collection efficiency. Further investigation needs to be done to determine the strength of vortex flow as this can produce a spinning motion that may wipe out the accumulated dust in the wall. It is desirable that the inlet velocity be kept constant and this can be achieved by inserting a porous plate at the inlet. Present study indicates that the inlet air flow is skewed at the entrance point but by varying the number of holes, arrangement and shape of porous plate it can be made more uniform.

References

[1] APTI, Air Pollution Training Institute (APTI) Course SI: 412B. 1998., in, U.S Environmental Protection Agency.,1998.
[2] M. HU, X. SUN, C. MA, Y. and LIU, L.-q. WANG, Numerical Simulation of Influence of Baffler in Electric Field Entrance to Form Skewed Gas Flow.
[3] L. Morawska, V. Agranovski, Z. Ristovski and M. Jamriska, 2002, 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 and N. Subaschandar, 2009 Flow simulation in an electrostatic precipitator of a thermal power plant, Applied Thermal Engineering, 29 (2009) 2037-2042.
[5] I. Gallimberti, 1998, Recent advancements in the physical modelling of electrostatic precipitators, Journal of Electrostatics,43 (1998) 219-247.
[6] S.M. Haque, M. Rasul, M.M.K. Khan, A. Deev and N. Subaschandar,2009,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.
[7] ANSYS FLUENT 12.0 Theory Guide.
[8] F. Dubois and W. Huamo, 2001, New advances in computational fluid dynamics—theory, methods and applications [M], in,Beijing: Higher Education Press,2001.