Applied Thermal Engineering 29 (2009) 2037–2042

Contents lists available at ScienceDirect

Applied Thermal Engineering

journal homepage: www.elsevier.com/locate/apthermeng

Flow simulation in an electrostatic precipitator of a thermal power plant

Shah M.E. Haque [a], M.G. Rasul [b,*], A.V. Deev [a], M.M.K. Khan [b], N. Subaschandar [a]

[a] Process Engineering and Light Metals (PELM) Centre, Faculty of Sciences, Engineering and Health, Central Queensland University, Gladstone, Queensland 4680, Australia
[b] College of Engineering and Built Environment, Faculty of Sciences, Engineering and Health, Central Queensland University, Rockhampton, Queensland 4702, Australia

ARTICLE INFO

Article history:
Received 17 October 2007
Accepted 21 October 2008
Available online 18 November 2008

Keywords:
Thermal power plant
Electrostatic precipitator
Turbulent flow
Flow distribution
CFD
Fluent

ABSTRACT

The performance of electrostatic precipitator (ESP) is significantly affected by its complex flow distribution arising as a result of its complex inside geometry. In the present study the gas flow through an ESP used at a local thermal power plant is modeled numerically using computational fluid dynamics (CFD) technique to gain an insight into the flow behavior inside the ESP. CFD code FLUENT is used to carry out the computations. Numerical calculations for the gas flow are carried out by solving the Reynolds-averaged Navier–Stokes equations coupled with the k–ε turbulence model equations. The results of the simulation are discussed and compared with on-site measured data supplied by the power plant. The predicted results show a reasonable agreement with the measured data. The model developed is a novel tool for the thermal power plant to predict the effect of possible modifications made to the ESP design on the flow pattern.

1. Introduction

Over recent years the particle emissions from process industries have been attracting more attention due to an anticipation of upcoming strict environmental protection agency (EPA) regulations. Industrial pollution can be controlled by energy recovery and conservation [1], replacing conventional industrial processes with continuous and energy efficient systems [2], or performance optimization of the emission control devices [3]. Electrostatic precipitators (ESP) are the most common, effective and reliable particulate control devices which are capable of handling large gas volumes with a wide range of inlet temperatures, pressures, dust volumes and gas conditions.

The performance of ESP is affected by the fluid flow characteristics inside this device wherein the shape, size and arrangement of collection electrodes, baffles, deflectors, etc. significantly influence the flow field. But it is very difficult to carry out detailed and reliable measurements of fluid flow inside an ESP as the geometry is very complex. CFD provides an alternative method, which is viable and less expensive to study the flow behavior inside the ESP. A suitable CFD model plays an important role in predicting the flow field characteristics and particle trajectories inside the ESP and optimizing flow distributions within the ESP by simulating proposed modifications. This ensures that the desired flow profiles are achieved, thus substantially reducing the outage time. However, only a limited number of research works could be found in the open literature for the prediction of turbulent flow behavior inside the ESP. Most of them are focused on 2D models based on simplified geometrical arrangements and ignored the effect of sudden expansion in geometrical configuration of an ESP. Zhao et al. [4] developed a simple 2D model which consists of a single discharge wire and two parallel plates. The 2D model developed by Skodras et al. [5] consists of three-wires and two parallel plates arrangements. Nikas et al. [6] simulated a 3D flow inside a laboratory scale precipitator of three-wire and two-plate arrangements. Varonos et al. [7] developed a 3D model and introduced smoothing grids to improve the flow characteristic of an ESP. But they simplified their model by introducing a porous region instead of creating any physical collecting plates in their CFD model. The numerical flow model of an ESP developed by Schwab and Johnson [8] replaced all the collection plates inside the ESP with equivalent resistance. Gallimberti [9] also used local loss coefficients in the governing equations to model the different wall profiles and other structures inside the ESP. Bottner and Sommerfeld [10] predicted turbulent flow in a test channel equipped with seven discharge wires. Dumont and Mudry [11] made a comparative study on flow simulation results obtained from different precipitator CFD models.

The above studies were broadly dedicated to simulate fluid flow inside the ESP with either simplified models or simplified geometries. The accurate aerodynamic characteristics of the flow inside an ESP in an operation may not be obtained without considering all of its major physical details. The novelty of this study is to develop a new 3D fluid flow model of a full scale ESP which considers all of its major physical features. It is to be noted that all the collecting electrodes (CE), baffles, gas deflectors, etc. are taken into account in this 3D model and have not been replaced by any

---

* Corresponding author. Tel.: +61 7 49309676; fax: +61 7 49309382.
E-mail address: m.rasul@cqu.edu.au (M.G. Rasul).

1359-4311/$ - see front matter © 2008 Elsevier Ltd. All rights reserved.
doi:10.1016/j.applthermaleng.2008.10.019

Nomenclature

$C_0, C_1, C_{3\varepsilon}$	constants
C_2	pressure jump coefficient = pressure loss coefficient per unit thickness (m^{-1})
g	gravity (m/s^2)
G_k	generation of turbulence kinetic energy due to the mean velocity gradients (m^2/s^2)
G_b	generation of turbulence kinetic energy due to buoyancy (m^2/s^2)
I	intensity
k	turbulent kinetic energy (m^2/s^2)
Δm	thickness of the perforated plate (m)
p	pressure (Pa)
Re	Reynolds number
U	velocity (m/s)
s_k, s_ε	user-defined source terms
S	modulus of the mean rate of strain tensor
u'	fluctuating velocity (m/s)
u_{avg}	average velocity (m/s)
Y_M	contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate

Greek symbols

α	permeability of the perforated plate (m^2)
Δ	differential
ε	turbulent dissipation rate (m^2/s^3)
η	strain
μ	dynamic viscosity ($N\,s/m^2$)
ν	kinematic viscosity (m^2/s)
ρ	density (kg/m^3)
σ_k	turbulent Prandtl numbers for k
σ_ε	turbulent Prandtl numbers for ε

Subscript

Dh	hydraulic diameter

equivalent porous region as other researchers have done in their studies. A detailed numerical approach and simulation procedure is presented to predict the flow behavior inside the ESP. The predicted results are compared with the on-site measured data. The flow model developed has the potential to better predict the effect of possible modifications and improvement in ESP design.

2. ESP geometry

The power station in this study has four power generating units of 350 MW capacity each. Each unit has two single-stage, plate-type, rigid-frame, cold-side and dry ESPs which are called pass A and pass B. Each pass has two gas-paths covering four zones as is shown in Fig. 1. The effective length, width and height of each casing are 30.36 m, 11 m and 13.1 m, respectively. The width and height of the CE walls are 5.76 m and 12.5 m, respectively. Each pass has 54 passages having 400 mm CE wall spacing. Discharge electrodes (DE) are welded into pipe frames with 2 frames per passage. The width of DE frame is 5.76 m and the heights are 5 m and 7.5 m. Rapping is the dust removal method for both collection electrodes and discharge electrodes. Three perforated plates with the thickness of 8 mm, 2 mm and 2 mm are located inside the inlet evase. The inlet evase is a pyramidal diffuser with large divergence angle (more than 50°) which is located in front of the rectangular collection chamber. The outlet evase, which is a convergent duct and located after the collection chamber, has a 2 mm thick screen inside it. Due to the symmetry in geometry only one-half of a gas-path has been modeled in this study.

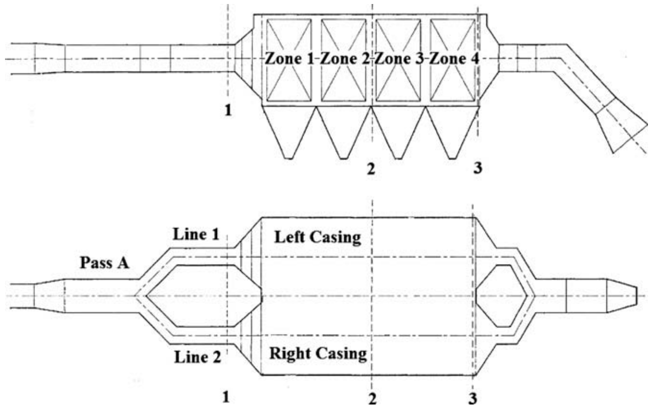

Fig. 1. Measurement planes for velocity distribution.

3. Numerical approach and simulation procedure

Numerical computation of fluid transport includes conservation of mass, momentum and turbulence model equations. The Fluent Inc. geometry and mesh generation software "Gambit" was used as a preprocessor to create the geometry, discretize the fluid domain into small cells to form a volume mesh or grid and set up the appropriate boundary conditions. The flow properties were then specified and the problems were solved and analyzed by "Fluent" solver.

3.1. Governing equations

The air inside the ESP was treated as incompressible Newtonian fluid due to the small pressure drop (<100 Pa) across the ESP. The flow was assumed to be steady and can be described by the conservation of mass equation:

$$\vec{\nabla} \cdot (\rho \vec{U}) = 0 \tag{1}$$

and the momentum equation:

$$\vec{U} \cdot \vec{\nabla} \vec{U} = -\frac{\vec{\nabla} p}{\rho} + \nu \vec{\nabla}^2 \vec{U} + \vec{g} \tag{2}$$

For the turbulent flow inside the ESP, the key to the success of CFD lies with the accurate description of the turbulent behavior of the flow. To model the turbulent flow in an ESP, there are a number of turbulence models available in Fluent. The realizable k–ε model is a relatively recent development and contains a new formulation for the turbulent viscosity and a new transport equation for the dissipation rate, ε which can be written as follows [12]:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M + S_k \tag{3}$$

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_j}(\rho\varepsilon u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \rho C_1 S\varepsilon - \rho C_0 \frac{\varepsilon^2}{k + \sqrt{\nu\varepsilon}} + C_{1\varepsilon}\frac{\varepsilon}{k} C_{3\varepsilon} G_b + S_\varepsilon \tag{4}$$