Applied Mathematical Modelling 000 (2016) 1–13

Contents lists available at ScienceDirect

Applied Mathematical Modelling

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

Multi-scale simulation of the gas flow through electrostatic precipitators

Xing-Lian Ye[a], Yin-Biao Su[a], Bao-Yu Guo[b,*], Ai-Bing Yu[c]

[a] Experimental and Research Centre, Fujian Longking Co., Ltd., Longyan 364000, China
[b] Department of Chemical Engineering, Monash University, VIC 3800, Australia
[c] Lab for Simulation and Modelling of Particulate Systems, School of Materials Science and Engineering, University of New South Wales, Sydney, NSW 2052, Australia

ARTICLE INFO

Article history:
Received 1 July 2014
Revised 17 May 2016
Accepted 16 June 2016
Available online xxx

Keywords:
Computational Fluid Dynamics
Flow distribution
Numerical simulation

ABSTRACT

The gas flow distribution significantly affects the collecting performance of an Electrostatic Precipitators (ESP) system. Computational Fluid Dynamics (CFD) method is here used to study the gas flow through the ESP system at different scales and in a coupled fashion. Firstly, numerical experiments are conducted at an orifice scale (unit cell study) and ESP unit scale (collection unit study) respectively, to determine the parameters of the perforated plates and ESP unit, such as the pressure drop coefficients. Flow distribution in the large scale ESP system is then simulated and adjusted based on a simplified network model. The simulation results and experiment data match well not only qualitatively but also quantitatively. The study shows that the multi-scale CFD approach is suited to predict the gas flow in ESP systems and the results provide useful guidance for ESP design and control.

1. Introduction

Electrostatic precipitators (ESP) is widely used, particularly in power plants for removing particulate matters from flue gas generated in coal fired boilers [1]. The structure of a typical ESP unit is complex, and comprises diffusers, collection chamber, electrical system, rapping system and solid transport system, etc. In practice, several individual ESP units are often combined in parallel or in series into a large system. So far, such an ESP system is the most common, effective and reliable particulate emission control device due to the merit of handling large volume of gases with a wide range of operation conditions, e.g., temperature, pressure, particle loading and chemical composition. The distribution of gas flow significantly affects the collecting performance of the ESP system, because the turbulent gas flow determines the trajectories of fine particles. Many of the earlier attempts to calculate the collection efficiency focused on the local behaviours of electric field and particles, with little attention to the complex gas flow [2–4]. In addition, understanding and controlling the gas flow are essential for further study of the complicated transport phenomena within the electric fields and optimisation of dust collection processes [5]. Traditionally, trial-and-error physical experiments are relied on to adjust the gas flow by installing guide baffles in the upstream pipework and extra distribution elements in the diffuser without really understanding the underlying fundamentals. With the advances in computer technology and numerical techniques, Computational Fluid Dynamic (CFD), particularly commercial CFD softwares are increasingly applied to the ESP [6]. Compared with in-house codes, these softwares are particularly convenient for dealing with complicated geometries.

* Corresponding author. Tel.: +61 3 99050844.
E-mail address: bob.guo@monash.edu (B.-Y. Guo).

http://dx.doi.org/10.1016/j.apm.2016.06.023
0307-904X/

Nomenclature

C_2	pressure loss coefficient
d	orifice diameter of perforated plate (m)
k	turbulence kinetic energy ($m^2\,s^{-2}$)
p	pressure (Pa)
S	source term
u,v	velocity ($m\,s^{-1}$)
α	permeability of the perforated plate (m)
β	porosity of the perforated plate
ε	turbulence eddy dissipation ($m^2\,s^{-3}$)
ρ	density ($kg\,m^{-3}$)
μ	dynamic viscosity (Pa s)
μ_t	turbulence eddy viscosity (Pa s)
ξ	pressure drop coefficient

Indeed, CFD has been used already to study the gas flows in various parts of an ESP system in recent years. Varonos et al. [1] used a modular approach to model gas flow and particle collection efficiency in an ESP. The velocity profile was adjusted by insertion of smoothing grids expressed by pressure loss coefficients. Noorpoor and Sadeghi [7] investigated the behaviour of airflow inside an electrostatic precipitator considering three different scenarios for distribution plates, i.e., without distribution plate, with a distribution plate after inlet duct and with two distribution plates (one after inlet duct and the other before outlet duct). Haque et al. [8–10] studied the flue gas flows in a laboratory ESP model and in a power station ESP using the CFD code Fluent, and compared the flow behaviour with experimental and on-site data supplied by the power plant respectively. Swamination and Mahalakshmi [11] numerically studied the flow through perorated plate in an ESP diffuser. Comparison of the simulation results with experimental data showed that the flow uniformity in the collection chamber depends on the location and porosity of the perforated plates. A too high porosity does not distribute the flow uniformly, whereas a too low porosity will effectively slow down the jet flow, but cause excessive pressure drop.

Bhasker [12] studied the flow numerically in ESP ducts with turning vanes by using several different flow solvers such as CFX-TASCFLOW, FLUENT, CFX, and STAR-CD to ensure meaningful predictions in the absence of measurement data. The simulation results suggest that the flow distribution can be improved by placing more turning vanes in the inlet duct.

The perforated plate, a key component in the diffuser, has been represented by an empirical correlation in nearly all the previous simulations. Hou et al. [13] divided the large scale ESP simulation into three steps, viz. unit cell of internal components, a single ESP unit, and entire ESP system. In the current paper, the above multi-scale method is revisited for further extension, application and validation. In particular, this approach is applied for the modelling of a large ESP system in a coupled fashion, which reduces the modelling assumptions (boundary conditions) that are otherwise necessary if the system is broken into manageable pieces. Here a large industrial ESP system (a network of two parallel ESP units) is considered, where the flow distribution is adjusted by various measures for optimisation purpose.

2. Governing equations and numerical method

The flue gas is assumed to incompressible and Newtonian. Thus, the steady-state governing equations include the conservation equations of mass, momentum and turbulence quantities, which are given as,

$$\frac{\partial u_i}{\partial x_i} = 0, \tag{1}$$

$$\frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left((\mu + \mu_t)\frac{\partial u_i}{\partial x_j}\right). \tag{2}$$

The standard $k-\varepsilon$ model was selected to simulate the turbulence [14].

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}. \tag{3}$$

$$\frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k - \rho\varepsilon + S_k, \tag{4}$$

and the eddy dissipation equation is given by,

$$\frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}G_k - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}G_k + S_\varepsilon, \tag{5}$$