Powder Technology 314 (2017) 577–588

Contents lists available at ScienceDirect

Powder Technology

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

Modelling of pollutant dispersion with atmospheric instabilities in an industrial park

Xiaofei Ma [a,b], Wenqi Zhong [a,b,*], Wei Feng [c], Gang Li [c]

[a] Key Laboratory of Energy Thermal Conversion and Control of Ministry of education, School of Energy and Environment, Southeast University, Nanjing 210096, PR China
[b] Centre for Simulation and Modelling of Particulate Systems, Southeast University - Monash University Joint Research Institute, Suzhou 215123, PR China
[c] Nanjing Sample Technology Group Co., Ltd., Nanjing 210049, PR China

ARTICLE INFO

Article history:
Received 8 May 2016
Received in revised form 19 August 2016
Accepted 26 August 2016
Available online 28 August 2016

Keywords:
Large Eddy Simulation
Plant canopy
Pollutant dispersion
Atmospheric instabilities
Industrial park

ABSTRACT

Large Eddy Simulation (LES) combined with a dynamic Smagorinsky subgrid scale model was used to simulate the wind field and particulate pollutant dispersion considering the effects of plant canopies and weather conditions. The model of the plant canopy was adopted to present the effects of drag force and buoyancy force and validated by bench mark results from previous reference. Then, using the model, four study cases in the Nanjing Sample Industrial Park were simulated under different weather conditions to consider the momentum effect of plant canopy, energy effect of plant canopy and the effect of atmospheric instabilities, respectively. The numerical results were also compared with those of idealized street canyon models, and it was proven that the accurate landform model was more practical to be applied in the study. The wind field and pollutant distribution were analyzed in each case to investigate the influence of plant canopy. It was shown that the wind velocity was lower near the canopy regions, especially in unstable conditions due to the entrainment effect of the buoyancy force. Meanwhile, the streamline could be changed by the buoyancy force in an upward trend. The distribution of the turbulent kinetic energy indicated that the turbulent flow in the atmosphere was weakened through the plant canopy, especially in unstable conditions. Considering the effect of buoyancy force, the direction of pollutant dispersion in horizontal profiles was different to the neutral condition, which proved that plant canopy can protect the downstream regions in unstable atmospheric conditions.

1. Introduction

Atmospheric pollutant has become one of the main threats to human health [1–3], especially in the regions around industrial parks. Industrial waste pollutants, such as pulverized coal and fly ash discharged from power plants and chemical plants, are the main types of atmospheric pollutant sources. The dispersion of particulate pollutants is greatly influenced by the surrounding landform and environment around the pollution source. However, the details and mechanism of how these factors affect the pollutant dispersion inside industrial parks still remain unclear. As a result, investigating pollutant dispersion inside an industrial park is becoming increasingly important in protecting human health around the park regions [4].

Currently, wind tunnel experiments, field measurements and Computational Fluid Dynamics (CFD) are the primary effective methods to investigate flow behaviors and contaminant dispersion in urban building arrays. Wind tunnel experiments have been an important method for studying the turbulent flow within canopy structures over past few decades [5,6]. Nevertheless, it is very difficult to manufacture the complex model of landforms in lab because of the high costs. Data received from actual field measurements are more accurate compared with wind tunnel experiments, however, resources for measurement points are very limited, making it impossible to obtain the data from every location inside an industrial park [7,8]. Compared with former two methods, CFD is more convenient and has lower costs regarding labour and material resources. Moreover, the detailed distribution of wind field and pollutant concentration can be obtained by CFD [9]. Therefore, CFD has been widely employed to investigate pollutant dispersion inside industrial parks.

The complex wind flow and pollutant dispersion inside industrial parks is typical of turbulent flow. Direct Numerical Simulation (DNS), Reynolds-Averaged Navier-Stokes equations (RANS) and Large Eddy Simulation (LES) are the commonly used methods to simulate the turbulent behavior. Among these methods, DNS is the most accurate for researching turbulent flows by directly resolving the Navier-Stokes equations. However, the efficiency of DNS can be very low when the computational domain is large due to the enormous cost associated with the computer source [10,11]. In contrast, the cost of RANS is the lowest, thus, it can be applied in high Reynolds number cases. However,

* Corresponding author at: Key Laboratory of Energy Thermal Conversion and Control of Ministry of Education, School of Energy and Environment, Southeast University, Nanjing 210096, PR China.
E-mail address: wqzhong@seu.edu.cn (W. Zhong).

http://dx.doi.org/10.1016/j.powtec.2016.08.062
0032-5910/

the accuracy of RANS is also lower than the other two methods due to many assumptions on the Reynolds stress [12]. Compared with DNS and RANS, the accuracy of LES is acceptable with a satisfactory computing time. In LES, the large-scale flow motion is obtained by solving the differential equations directly, and the influence of small-scale motion acting on the large-scale motion is evaluated by subgrid models [13–15]. In this study, the computational domain of the actual industrial park was very large so that the DNS cannot be used due to the large requirement of computer source. Meanwhile, the simulation results of pollutant distribution should be sufficiently accurate to capture the main features of the fluid flow behavior so that the RANS is not appropriate because of the low accuracy of the simulation results. Thus, LES is adopted.

The pollutant in air commonly exists as solid particles and the dispersion of particulate pollutant can be simulated by using Eulerian-Lagrangian simulation method. Liu et al. [33] studied the particle dynamic behavior in the exhaust gas in the near-wake region behind the studied ground vehicle using large-eddy simulation (LES) with the aerosol dynamics and dispersion model based on direct quadrature method of moments (DQMOM) approach. Morikone et al. [34] proposed a Lagrangian approach using LES to simulate the pollutant dispersion, the continuous phase is determined by large-eddy simulation whereas the dispersed phase is simulated in a Lagrangian approach. The results of the study indicate that the particle shape and size influences the particle dispersion.

CFD studies on idealized street canyon models have been popular for a couple of years, these can be regarded as a preliminary and less costly approach to capturing the general features of turbulent flows around buildings at relatively high Reynolds number. Hanna et al. [16] investigated the mean flow and turbulent statistics within simple obstacle arrays via CFD and discovered that the layout of arrays has a substantial influence on the flow field. The wind field and pollutant dispersion in a building array model were studied by Shi et al. [17] where the regularity of pollutant distribution in idealized building array is concluded. Wang and Cui [18] researched the unstable stratified turbulent flow over urban-like building arrays and analyzed the flow behaviors inside the canyon gap. Regularities of wind flows and pollutant dispersion were found in the previous researches in idealized models of street canyons. Nevertheless, the difference of terrain, building structures and building locations can greatly affect the wind field and pollutant distribution [19]. Therefore, it is necessary to discuss the wind flow in real landform models with various building structures and canopies to more accurately simulate the flows and contaminant distributions.

It is also important to consider the influence of plant canopies because drag and buoyancy forces can be generated by plant canopies under different atmospheric conditions. Therefore, the model of plant canopy is indispensable to more accurately simulate pollutant dispersion in an industrial park. Gu et al. [20] created an idealized model of street canyon with tree planting to investigate the wind flows inside. Gromke and Ruck [21] concluded that different aspect ratios of plant canopies have a marked influence on the pollutant concentration in street canyons. Ries and Eichhorn [22] utilized the plant canopy model developed by Shaw and Schumann [23] to investigate flows and contaminant distributions in street canyons with plant canopy and found that the variation in the results can be approximately 10% under effects of plant canopy. Apart from the hindering effect, the instability of weather condition can also change the distributions of wind and pollutants because the canopy can absorb solar radiation and change the temperature distributions locally. Xie et al. [24] studied the effect of solar radiation on pollutant distribution in street canyons with different structures and found that the flow structure could be completely different under solar heating. In conclusion, the simulation of idealized street canyon models can provide general regularities of wind flow and pollutant distribution and the effects of plant canopies under given weather conditions is essential when examining flow behaviors. However, the investigations of the plant canopy influence on the wind field and pollutant dispersion in real industrial parks are scarce.

In this study, a real terrain model with crowded buildings and plant canopies of Nanjing Sample Industrial Park was constructed to study the wind flows and contaminant dispersion under different unstable weather conditions. Numerical simulations were conducted using LES along with dynamic Smagorinsky subgrid scale model [25]. The wall function was applied to make sure the non-slip condition near walls. The momentum and energy sources were included in the physical model of plant canopy, and the entire model was validated by comparison with bench mark results from previous studies. Then, four different cases under various weather conditions were simulated using the plant canopy model to explore the effects of plant canopy on wind field and pollutant dispersion in a real relief model.

2. Simulation method

2.1. Governing equations

In this study, the wind flow with contaminants among building arrays could be regarded as a low-speed atmospheric motion that satisfies the conditions of incompressible Navier-Stokes equations. The particulate pollutant is assumed to be the continuous phase and the dispersion motion can be described by the transport equations. The LES method should adopt the filter method to separate different scales of vortices. First, large-scale turbulent motion is obtained by directly solving the differential equation. The influence that small-scale motion acting on the large-scale motion is simulated using subgrid models. The governing equations of this study are written as follows

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

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\rho \partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j} + \frac{\partial \tau_{ij}}{\partial x_j} + f_i \tag{2}$$

Eqs. (1) and (2) are the continuity and momentum equations, respectively. In Eq. (1), u_i is the resolved velocity and x_i is spatial coordinate. In Eq. (2), t is the time and p is the pressure, ρ represents density of material andν represents the dynamic viscosity, τ_{ij} is the subgrid-scale stress tensor which represents the impact of small-scale vortexes to large-scale ones and f_i stands for the external force, such as the drag force of plant canopy and buoyancy force due to the difference of air density.

In addition to Eqs. (1) and (2), the contaminant transport and energy equations can be described as follows

$$\frac{\partial c}{\partial t} + u_j \frac{\partial c}{\partial x_j} = D \frac{\partial^2 c}{\partial x_j \partial x_j} + \frac{\partial T_{cj}}{\partial x_j} + q_c \tag{3}$$

$$\frac{\partial T}{\partial t} + \frac{\partial (u_j T)}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\mu \frac{\partial T}{\partial x_j} + \tau_{Tj}\right) + S \tag{4}$$

where c is the pollutant concentration, D is the molecular mass, T_{cj} is the subgrid scale mass flux and q_c represents the intensity of contaminant source in Eq. (3). In Eq. (4), T is temperature, μ is the thermal diffusivity, τ_{Tj} is the subgrid thermal flux and S is the heat source which is imparted to the air from plant canopy.

2.2. Subgrid scale model

The subgrid scale stress tensor τ_{ij} as well as the subgrid scale flux T_{cj} and τ_{Tj} is unknown quantity in Eqs. (2), (3) and (4) that are required to be closed using the subgrid scale model of turbulent flows. Here, we