Amirkabir Journal of Mechanical Engineering

Amirkabir J. Mech. Eng., 53(5) (2021) 669-672
DOI: 10.22060/mej.2020.17685.6656

Modeling of Turbulent Atmospheric Boundary Layer and Dispersion of Solid Pollutant Particles in an Urban Area Using Large Eddy Simulation

M. K. Moayedi[*] , V. Azaditalab

Department of Mechanical Engineering, University of Guilan, Rasht, Iran.

ABSTRACT: In this paper, the air flow field around buildings is simulated to predict the dispersion of the fine solid pollutants. The large eddy simulation approach has been used to model the turbulence flow. In the first part of this research, a simple model including a building has been simulated and the obtained results are compared and validated with the experimental data obtained from a wind tunnel test. By setting the optimal parameters of the numerical model in the first part, in the second part, an area of Tehran city with high-rise buildings and irregular urban layout is considered and the velocity field and deposition of the contaminant particles in this model are also simulated. The result obtained in the first part of show good agreement with the experimental data and in both models the effect of some variables like the arrangement of buildings (in urban model) and the wind velocity are investigated. To analyze the value of the pollutant concentration in the urban area at each time, the integral of this variable on some important surfaces has been calculated over time and the effect of urban area layout on the integration is discussed.

Review History:

Received: Jan. 12, 2020
Revised: May. 11, 2020
Accepted: Jun. 20, 2020
Available Online: Oct. 15, 2020

Keywords:

Large eddy simulation
Air pollution
Computational fluid dynamics
Particulate material pollutant
Environmental fluid dynamic

1. Introduction

The first step for understanding the pollutant dispersion in an urban area by the computational fluid dynamics is to model the wind comfort pedestrian in that area. The nature of wind flow in such problems is turbulent and it must be considered in accurate engineering problems. For this purpose, there are two options, first to calculate this turbulent flow directly in direct numerical solutions and the second to model that by the use of turbulent flow modeling methods. The first options need a huge amount of calculations resources and usually does not appropriate for engineering problems so that the second option is most useful in engineering problems such as wind comfort pedestrian. Also to model the turbulent flow field there are two approaches, one of them is the Reynolds Average Naiver-Stocks (RANS) approaches which are not time dependent solutions and can not analyses the variations of flow parameters over time but the second approaches like Large Eddy simulations (LES) uses the mass averaging form of the governing equations and also has good accuracy and it is between the DNS and RANS methods by the reasonable use of calculations resources.

2. Large Eddy Simulation

LES filters a flow field in terms of the scale size of eddies and resolves the governing equations directly for large eddies. If the filter width is equal to the grid size, the filtered incompressible governing equations are:

$$\frac{\partial \overline{u_i}}{\partial t} = 0 \tag{1}$$

$$\frac{\partial \overline{u_i}}{\partial t} + \overline{u_j}\frac{\partial \overline{u_i}}{\partial x_i} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \nu \frac{\partial^2 \overline{u_i}}{\partial x_j^2} - \frac{\partial \tau_{ij}}{\partial x_j} \tag{2}$$

Here, the overbar indicates spatial filtering and not time-averaging as in RANS. Therefore, and are the filtered velocity and pressure, respectively. Additional tensor terms, are introduced due to the filtering operation (analogous to the Reynolds stresses resulting from Reynolds-averaging) and are commonly termed as the SubGrid-Scale (SGS) stresses:

$$\tau_{ij} - \frac{1}{3}\tau_{kk}\delta_{ij} = -2\mu_t \bar{S}_{ij} \tag{3}$$

$$\mu_t = \rho L_s^2 \bar{S} \tag{4}$$

Here is the subgrid-scale turbulent viscosity, an is the rate of strain tensor for the resolved scale [1].

3. Particle Motion Model

Using the Lagrangian approach, the force balance for each solid fine particle is as follows:

$$\frac{d\vec{u}_p}{dt} = \frac{\vec{u} - \vec{u}_p}{\tau_r} + \frac{\vec{g}(\rho_p - \rho)}{\rho_p} + \vec{F} \tag{5}$$

*Corresponding author's email: moayyedi@qom.ac.ir

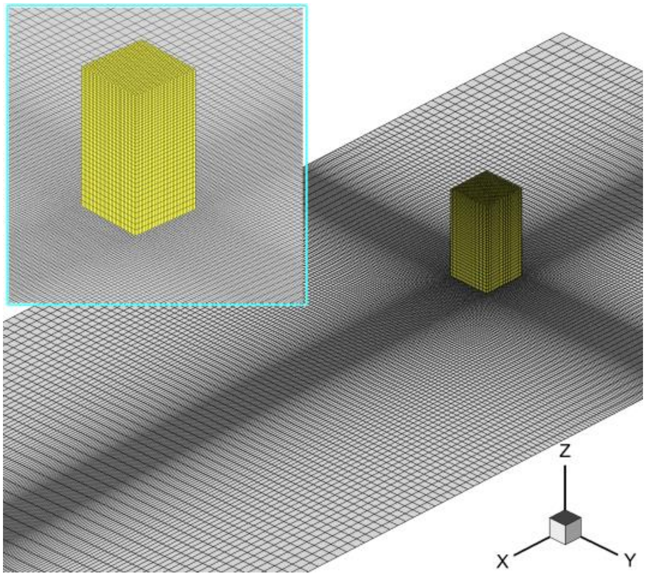

Fig. 1. Computational grid for benchmark model

In this regard, the values with the subtitle p correspond to the particle and the force F is the result of external forces on the particle, which here is simply the force of Saffman,

4. Computational Model Description

In the first part of this research, a high rise building with the 2:2:1 aspect ratio has been chosen which is experimentally modeled in the wind tunnel by Meng and Hibi [2]. It has been considered as a benchmark model to evaluate the optimal parameter in CFD modeling by using Ansys Fluent commercial software. The results of this modeling were validated with the experimental data for the flow field parameters. This model and the grid are illustrated in Fig. 1.

In the second part, an area of Tehran city with high-rise buildings and irregular urban layout is considered and the velocity field and the dispersion and deposition of the contaminant particles in this real sample environment are also simulated. This model and the related computational grid shown in Fig. 2.

5. Results and Discussion

For the modeling of both problems, firstly the flow field has been simulated for a sufficient period of time and then assumed that a specific amount of solid fine particle has been entered to the domain for a short period of time and the movement of these particles has been tracked until they abandoned the domain and the integral of pollutants concentration on some important surfaces calculated over the time. Fig. 3 shows the contour of the mean velocity of the flow at a plane on Y=0 and Z=0.01 m for the bench model. Fig. 4 shows this parameter at Z=0.01 m for the real urban model.

To evaluate the effect of important variables on the pollutant dispersion some other simulations for both models have been performed. For example, to evaluate the effect of wind velocity on the pollutant dispersion in the bench model three test cases with different wind velocities at the inlet (C02: High, C03: Medium, C04: Low Velocity) assumed and simulated. In Fig. 5 integral of pollutants concentration at the ground surface over the time for these three cases has been compared.

To evaluate the effect of the building complex on the deposition of pollutants downstream of flow the surface of the ground in the real urban model has been divided into three sections. There is no building in the first section and almost all of the buildings exist in the second one and the third part in downstream area of model and the three sections are equal. Fig. 6 shows the variation of the integration of the pollutant concentration on these three sections.

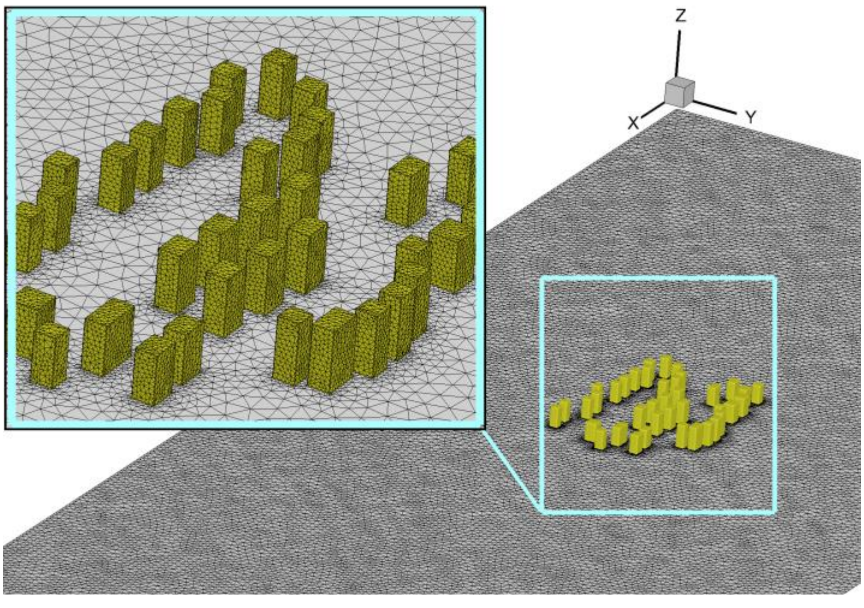

Fig. 2. Computational grid in real model

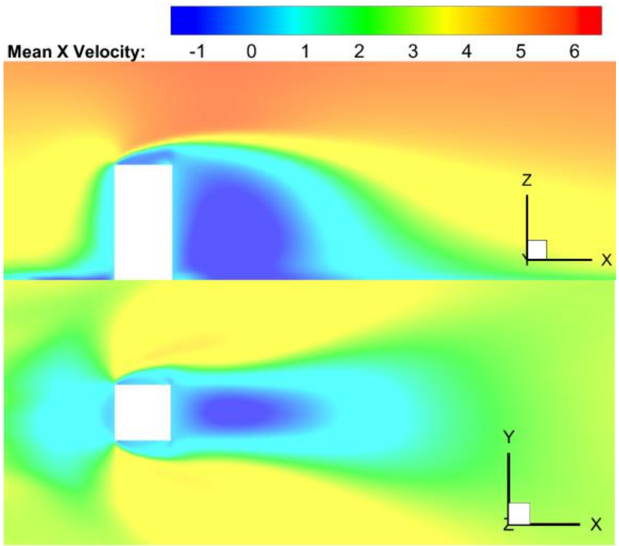

Fig. 3. Contour of mean X-Velocity in y=0 plane (upper) and z=0.01m plane (lower)

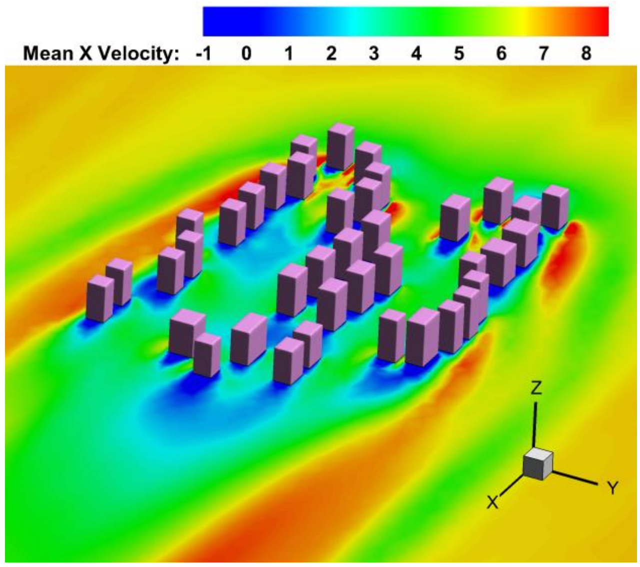

Fig. 4. Contour of mean X-velocity at z=1cm plane

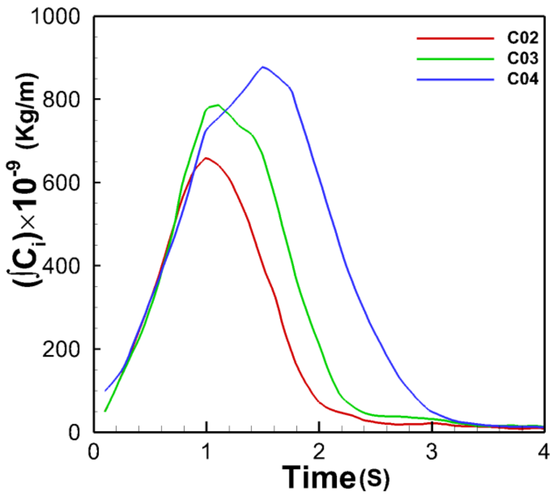

Fig. 5. Variations of the integral of concentration of pollutants on the ground surface versus time for three test cases

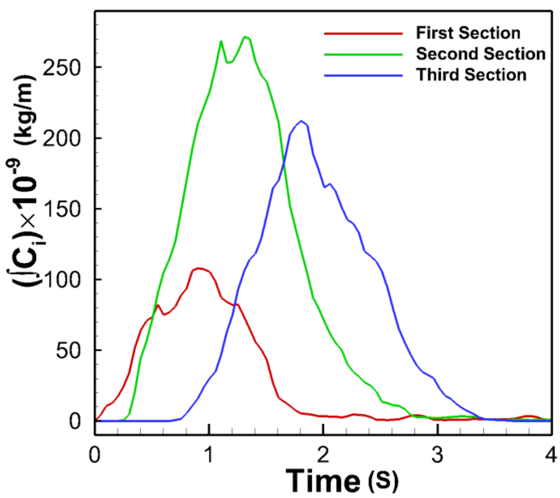

Fig. 6. Variations of the integral of concentration of pollutants on three parts of ground surface

6. Conclusions

In the first part of this study results showed good agreement between CFD and wind tunnel result and in the second part, this approach has been extended for a complicated domain with further analysis like studying the dispersion of solid fine pollutants. The ability of the LES method to manifestation of small-scale characteristics of the flow over the time has been represented, and by considering the lower cost of LES rather than the DNS simulations in computational resources, this study shows that in order to analysis the main topics in the environmental fluid dynamics this approach is appropriate. The results for calculation of concentration of pollutants shows that urban development and construction of high-rise buildings can affect the air flow pedestrian and contamination of pollutant particles in the downstream of the high-rise buildings and this has a harmful effect on human health and other well-beings so to predict this, the urban engineers can use the CFD tools to locate buildings with the optimum shape and height.

References

[1] [1] A.Q.C. Company, Tehran Annual Air and Noise Quality Report, QM98/02/01(U)/1, 2019.

[2] [2] Y. MENG, K. HIBI, Turbulent measurments of the flow field around a high-rise building, Journal of Wind Engineering & Industrial Aerodynamics, 1998(76) (1998) 55-64.

HOW TO CITE THIS ARTICLE

M. K. Moayedi, V. Azaditalab, Modeling of Turbulent Atmospheric Boundary Layer and Dispersion of Solid Pollutant Particles in an Urban Area Using Large Eddy Simulation, Amirkabir J. Mech. Eng., 53(5) (2021) 669-672.

DOI: 10.22060/mej.2020.17685.6656