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

Mohammad Kazem Moayedi[1*], Vahid Azaditalab[2]

[1&2] CFD, Turbulence and Combustion Research Lab.
Department of Mechanical Engineering, University of Qom, Iran

ABSTRACT

In this paper the air flow field around the building 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 include a building has been simulated and the obtained results are compared and validated with the related experimental data obtained from wind tunnel. 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 at the first part of show good agreement with the experimental data and in the 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 the time and the effect of urban area layout on the integration is discussed.

KEYWORDS

Large Eddy Simulation (LES), Air Pollution, Computational Fluid Dynamics (CFD), Particulate Material Pollutant (PM), Environmental Fluid Dynamic.

[*] Corresponding Author: Email: moayyedi@qom.ac.ir

1. Introduction

The first step for understanding the pollutant dispersion in an urban area by the computational fluid dynamic 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 depended solutions and can not analyses the variations of flow parameters over the time but the second approaches like Large eddy simulations (LES) uses the mass averaging form of the governing equations and also has a good accuracy and in fact it is between the DNS and RANS methods by the reasonable use amount 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:

$$\frac{\partial \bar{u}_i}{\partial t} = 0 \tag{1}$$

$$\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_i \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \bar{p}}{\partial x_j} + \nu \frac{\partial^2 \bar{u}_i}{\partial x_j^2} - \frac{\partial \tau_{ij}}{\partial x_j} \tag{2}$$

Here, the overbar indicates spatial filtering, and not time-averaging as in RANS. Therefore, $\bar{u}_i$ and $\bar{p}$ 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 μ_t is the subgrid-scale turbulent viscosity, and S_{ij} is the rate of strain tensor for the resolved scale[1].

3. Particle Motion Model

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

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

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

At the first part of this research a high rise building with the 2:2:1 aspect ratio 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 validated with the experimental data for the flow field parameters. This model and the grid illustrated in fig.1.

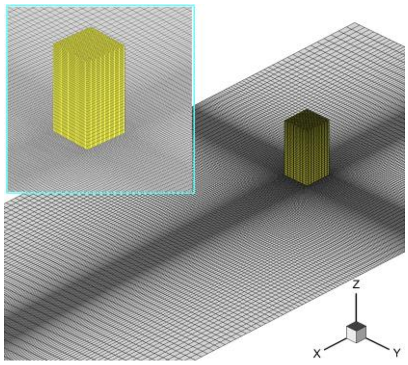

Fig1. Computational grid for benchmark model

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.

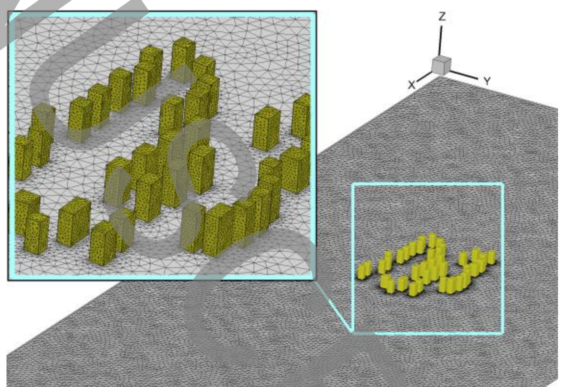

Fig2. Computational grid in real model

5. Results and Discussion

For the modeling 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 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.

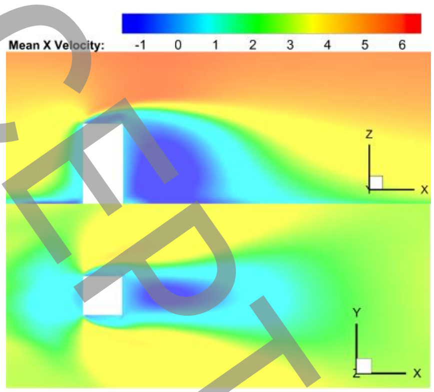

Fig3. Contour of Mean X-Velocity in *y*=0 Plane, B: In *z*=0.01 m Plane

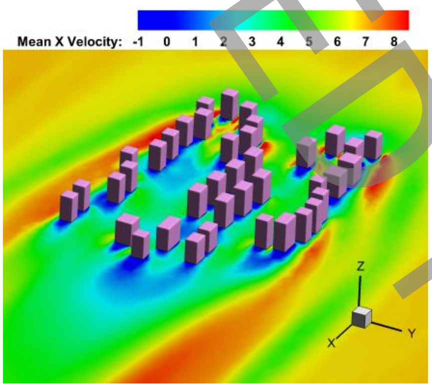

Fig4. Contour of mean X-velocity at *z*=1cm plane.

To evaluate the effect of important variables on the pollutant dispersion some other simulations for both models has 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 velocity at the inlet (C02: High, C03: Medium, C04: Low Velocity) assumed and simulated. In the 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 building complex on the deposition of pollutants at the downstream of flow the surface of ground in the real urban model has been divided in three sections. There is no building in the first section and almost all of buildings exist in the second one and the third part in in the downstream area of model and the three sections are equal. Fig.5 shows the variation of the integration of the pollutant concentration on these three sections.

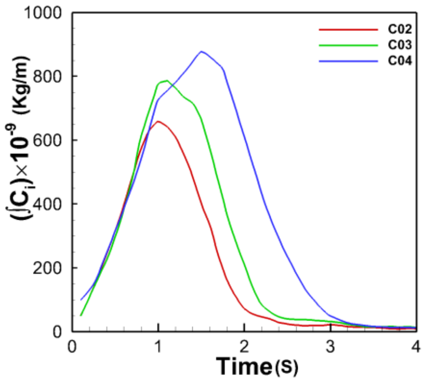

Fig5. Variations of the integral of concentration of pollutants on the ground surface versus time for three test cases.

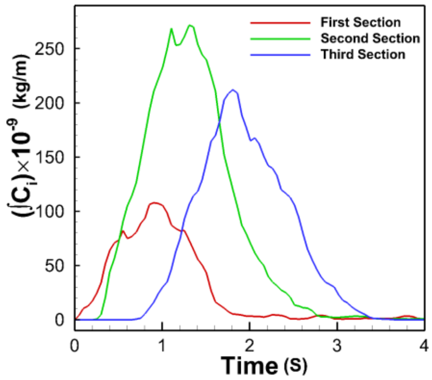

Fig6. Variations of the integral of concentration of pollutants on three parts of ground surface

6. Conclusion

Large eddy simulation approach has a good accuracy for modeling the solid pollutants deposition in the urban areas and can be used to make decisions about the expansion of urban planning to avoid the pollution density. By using CFD method and the large eddy simulation approach all flow field variables in at any point of the domain can be calculated over the time with good relative accuracy and the cost of simulation is very affordable rather than the experiment or the DNS solutions.

7. References

[1] S.M. Salim, S.C. Cheah, A. Chan, Numerical simulation of dispersion in urban street canyons with avenue-like tree plantings: comparison between RANS and LES, Building & Environment, 46(9) (2011) 1735-1746.

[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.