International Journal of Civil and Structural Engineering– IJCSE
Volume 2 : Issue 1 [ISSN : 2372-3971]
Publication Date: 30 April, 2015

CFD Simulation on Ventilation Performance in Residential Community with Multi Building Layouts

[Cui-e DUAN[1,2], Zhaolin GU[1], Jianying JIAO[1], Wei-Zhen LU[2,*]]

Abstract — **This paper reports a numerical simulation of three-dimensional air flow around different obstacles (buildings). The results can be used for assessing the natural ventilation performance and the potential impact on pollutant dispersion.**

Keywords — **Ventilation, LES model, Residence, Drag source term, Tunnel effect**

I. Introduction

With rapid increase of urban population and fast expansion of urban area in metro-to-mega cities, urban atmospheric environmental quality addresses extensive attention and plays key role to public health. Turbulent flows around three-dimensional obstacles (buildings) are common in nature and engineering applications. In recent years, with the rapid increase of computer property, numerical simulation technology is becoming a very effective method to study complex turbulent flows. The large eddy simulation (LES) is one of the most important methods to study complex turbulent flows. LES can capture the turbulent and average characteristics of flow, while has higher accuracy than RANS. LES also has smaller calculation task than DNS [1-7].

A commonly used SGS eddy viscosity model is Smagorinsky model (SM) which the sub-grid viscosity is associated with the norm of the resolved strain rate tensor which is large scale, and the filter cutoff length which is not the intrinsic mixing length associated with the subgrid scale (SGS). The simulations results show that SM produces excessive dissipation near the wall [4-7]. To improve SM, Several scholars proposed scale similarity model assuming that the statistical structure of the tensors constructed on the basis of sub-grid scale is similar to that of their equivalents evaluated on the smallest resolved scales but suffering fluctuations of results [8-14]. In order to improve the instability of DSM in simulation, several SM models were proposed to tackle the drawbacks of existing SM models. Gu et al. proposed a new SGS model and successfully applied to the channel flow simulation, which was entitled the universal sub-grid stress model (USM) [15-17].

In this work, the USM model is introduced to improve the prediction accuracy of wind field and pollution dispersion around different obstacle arrays. The geometries of the building layout and flow parameters are taken from the published wind tunnel experiments by Davidson et al. [18]. Subsequently, the results of the USM model are compared with published experiments, to show the promise of the USM model in simulating the complex turbulence flow and high Reynolds number turbulence flow.

II. Model development

There are two core standpoints in USM proposed by analyzing the similarities of the eddy viscosity formulations for LES and RANS. Firstly, the sub-grid eddy viscosity is relates indeed to the norm of the strain rate tensor of the smallest resolved scales, while the sub-grid tensor is still associated with the strain rate tensor of large scales like in the classical Smagorinsky model. It is verified by the direct numerical simulations (DNS) results of channel flows with different Reynolds numbers which show that the near-wall turbulence is associated not only with the fluctuation of small scales, but also with the fluctuation of large scales. The same prove was fond in RANS which shows that the eddy viscosity of the k-ε mode is related to the turbulence energy and dissipation rate of fluctuations.

The proposed universal SGS eddy viscosity model (USM) is described as follows:

$$\tau_{ij} = -2\nu_{sgs}\bar{S}_{ij} + \delta_{ij}\tau_{kk}/3 \tag{1}$$

Where is $\bar{S}_{ij}$ the resolved strain rate tensor which is expression by the large scales? The expression formula is described as follows:

$$\bar{S}_{ij} = (\partial\bar{u}_i/\partial x_j + \partial\bar{u}_j/\partial x_i)/2 \tag{2}$$

However, the sub-grid eddy viscosity ν_{sgs} is expression by the smallest scale. The expression formula is described as follow:

Cui-e DUAN, Zhaolin GU, Jainying JIAO
School of Human Settlements and Civil Engineering, Xi'an Jiaotong University
Xi'an, Shaanxi, PRC

Wei-Zhen LU
Department of Civil and Architectural Engineering, City University of Hong Kong
Hong Kong, HKSAR *

$$\nu_{sgs} = l_{sgs}^2 \left|\overline{S}^{\nu}\right| \tag{3}$$

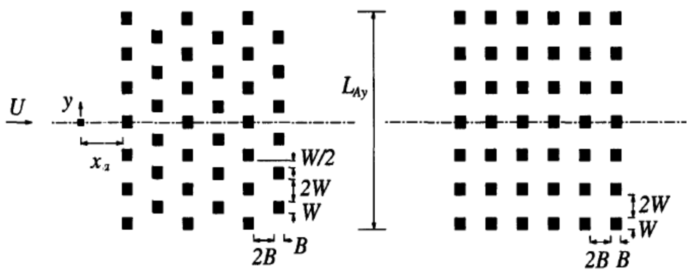
Where it is the module of smallest strain rate tensor which is described as follows:

$$\left|\overline{S}^{\nu}\right| = (2\overline{S}_{ij}^{\nu}\overline{S}_{ij}^{\nu})^{1/2} \tag{4}$$

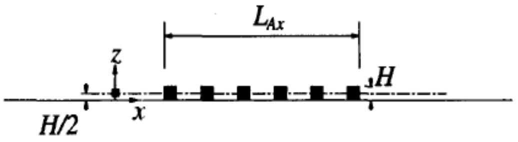
$$\overline{S}_{ij}^{\nu} = (\partial(\overline{u}_i - \overline{\overline{u}}_i)\big/\partial x_j + \partial(\overline{u}_j - \overline{\overline{u}}_j)\big/\partial x_i)\big/2 \tag{5}$$

Where $\overline{u}_i$ and $\overline{\overline{u}}_i$ are the resolved velocities in the sense of the grid filter and the test filter, respectively.

In order to simulate the wind tunnel experiment by Davidson et al, non-uniform grids of the computational domain are used for mesh generation of the staggered and aligned obstacle arrays. The grid number of the staggered obstacle array is 160×170×30 in stream-wise, transversal and vertical directions. The geometrical configuration is shown in Fig. 1.



a) Top plan views of the staggered array configuration (on the left) and the aligned array configuration (on the right)

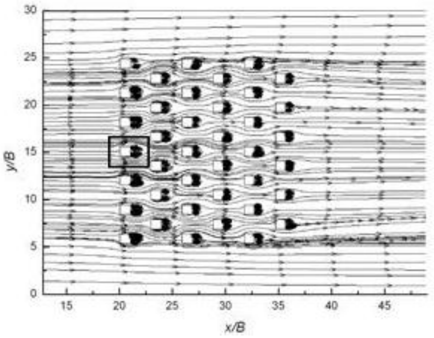


b) Side view of the aligned array configuration

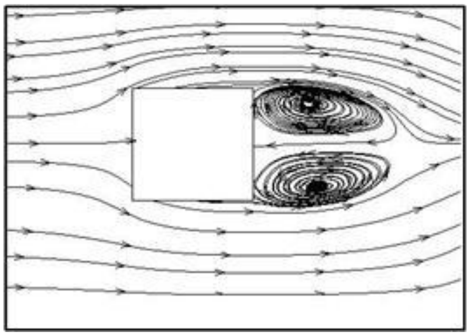
Fig. 1 Geometry configuration of building arrays

III. RESULTS AND DISCUSSION

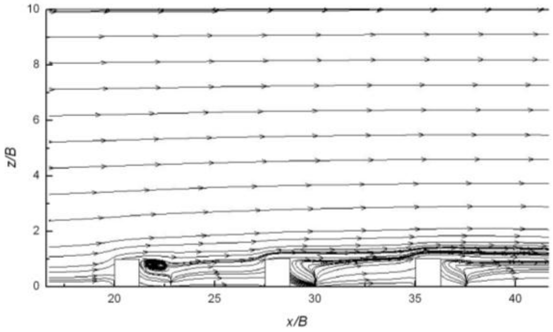
In order to validate the USM presented here, numerical simulation of the staggered obstacle array is carried out. Experimental data of Davidson et al. and numerical results of Shi et al. are then compared with our numerical result to validate the new SGS modelling. Fig. 2 shows the flow patterns of air passing through the staggered obstacle array. Horizontal streamlines of mean velocity at z = H/2 are presented in Fig. 2a. In order to make a clear illustration of the flow field, we make a magnification of the flow field near the center of the first row obstacles, as shown in Fig. 2b. We also make a vertical cut along the center line of the simulation domain, and the streamlines in this face is shown in Fig. 2c. It can be clearly observed that behind the first building there are vortexes, which indicates that in this domain there is a recirculation zone. Fig. 2c also shows that the flow is divergent at the front row and convergent behind the last row, which is in agreement with the findings in Davidson et al.'s work.



a) Horizontal profiles of the u-component of mean velocity at the height, z = H/2



b) Local streamlines around single block



c) Vertical profiles of the u-component of mean velocity at the center line of the staggered obstacle array

Fig. 2 Flow patterns of the staggered obstacle array

IV. CONCLUSIONS

The rapid urbanization in the past decades has gradually changed the traditional physical sense of urban environment. Concerning the continuing population expansion and land resource shortage, the condensed, high-rise building jungle presents the distinct character in most metropolitan cities and, meanwhile, leads to bad ventilation situation in the local climate, which causes adverse impact on residents' health. In this paper, wind field and contaminant dispersion in two kinds of building array geometries are simulated using the modelling. The wind tunnel experimental data is used to validate the modelling. The relative errors of lateral profiles of u-components of mean velocities of flows at the last row of the staggered obstacle array and the aligned obstacle array at the half height of the obstacle (building) along with the stream-wise are 15.2% and 8.6%, respectively, while the error of Shi et al.'s [19] simulation results at the staggered obstacle array along with the stream-wise is 46.7%. Velocity fluctuations of USM at the staggered and aligned obstacle arrays are in agreement with those of experiment. The results indicate that the USM can make a more accurate prediction of the mean velocity and velocity fluctuations. The USM is highly suitable for the simulation of the multi-scale turbulent flow of city underlying surface, the high Reynolds number turbulent flow and the complex turbulent flow.

Acknowledgment

The work was partially supported by Hong Kong RGC-GRF Grant (CityU 118212), Strategic research grant, City University of Hong Kong [Projects No. CityU-SRG 7002718] and National Natural Science Foundation of China [No. 10872159].

References

[1] Piomelli, U. (1993) "High Reynolds number calculations using the dynamic subgrid-scale stress model," Phys. Fluids A, 5: 1484-1490

[2] Piomelli, U., and Junhui. L. (1995) "Large eddy simulation of rotating channel flow using a localized dynamic model," Phys. Fluids, 7: 839-848

[3] Durbin, P. A. (1991) "Near-wall turbulence closure modeling without damping function," Theor. and Comput. Fluid Dynamics, 3: 1

[4] Speziale, C. G., and Ngo, T. (1988) "Numerical solution of turbulent flow past a backwards-facing step using a nonlinear k-ε model," International Journal of Engineering Science, 26: 1099

[5] Kim, j., Moin, P., and Moser, R. (1987) "Turbulence statistics in fully developed channel flow at low Reynolds number." J. Fuild Mech., 177: 133-166

[6] Moser, R. D., Kim, J., and Mansour, N. N. (1999) "Direct numerical simulation of turbulent channel flow up to R_e=590", Phys. Fluids, 11(4): 943-945

[7] Smagorinsky J., (1963) Mon. Weather Rev., 91: 99.

[8] Bardina J., AIAA, 80(1980) 1357.

[9] Germano M., Piomelli U., Moin P. and Cabot W.H. (1991) Phys. Fluids A, 3.

[10] Sagaut P. (2005) Large Eddy Simulation for Incompressible Flows (Springer, Berlin).

[11] Hughes T. J. R., Mazzei L. and Jansen K. E., (2000) Comput. Vis. Sci., 3: 47.

[12] Jeanmart H. and Winckelmans G., (2007) Phys. Fluids, 19, 055110.

[13] John V. and Kindl A., J. Comput. Phys., (2010) 229: 301.

[14] Iwamoto K., Kasagi N. and Suzuki Y. (2005) Proceedings of the 6th Symposium on Smart Control of Turbulence, Tokyo.

[15] Gu ZL, Jiao JY, Zhang YW, Su JW. (2011) "Large eddy simulation using a dynamic mixing length". Int J Numer Meth Fluid, DOI: 10.1002/fld.2645.

[16] Gu ZL, Jiao JY, Su JW, Zhang YW. (2011) "The nature of a universal subgrid eddy viscosity model in a turbulent channel flow". EPL, DOI: 10.1209/0295-5075/94/34003.

[17] Gu ZL, Jiao JY, Zhang YW. (2009) "A new subgrid scheme based on dynamic mixing length for turbulent flows", International Symposium on Multi-Phase Flow in Atmospheric Boundary Layer: Wind Erosion, Dust Storms, Sand Saltation, Snow Drift, Lanzhou, China, pp 16-19.

[18] Davidson MJ, Snyder WH, Lawson Jr, Hunt CR. (1996) "Wind tunnel simulations of plume dispersion through groups of obstacles", Atmos Environ, 30: 3715-731.

[19] Shi RF, Cui GX, Wang ZS, Xu CX, Zhang ZS. (2008) "Large eddy simulation of wind field and plume dispersion in building array. Atmos Environ", 42: 1083-1097.

About Author (s):

Image

Prof. Lu joined City University of Hong Kong in 1996 and currently serves as a Professor in Dept. of Architecture and Civil Engineering. Her main research interests include: indoor air quality, air pollution and assessment, built environmental assessment, wind effect on high-rise buildings, application of Computational Fluid Dynamics (CFD) and soft computing in environmental engineering.