Sixth International Symposium on Turbulence and Shear Flow Phenomena
Seoul, Korea, 22-24 June 2009

LARGE EDDY SIMULATION OF WIND FIELD AND TRAFFIC POLLUTANT DISPERSION IN A RESIDENCE AREA

Guixiang CUI
Department of Engineering Mechanics,
Tsinghua University
Beijing, 100084, China
cgx@tsinghua.edu.cn

Ruifeng SHI and Chunxiao XU and Zhaoshun ZHANG
Department of Engineering Mechanics,
Tsinghua University
Beijing, 100084, China

Zhishi WANG
Faculty of Science and Technology
Macau University
Macao 9999078, China

ABSTRACT
In this paper the traffic pollution in downtown Macao is investigated by Large Eddy Simulation (LES). The complex building layout is simulated by a composite model that the buildings and streets are properly resolved with fine grids in the residential area where the wind field and traffic pollution are most concerned while roughness element model is used for buildings in the surrounding area with coarse grids. The wind field and carbon oxide distribution produced traffic pollution are computed from 7am to 5pm in an autumn day. The numerical results are checked by wind tunnel measurement and field observation with satisfaction. The detailed distribution of traffic pollution in street network is revealed and the influence of building configurations on the pollutant dispersion is analyzed.

INTRODUCTION
Acceleration in urbanization and rapid growth of mobile vehicles result in serious problem of air quality in urban residence areas. The major tools for investigation of city air quality are field observation, laboratory model test and numerical simulation. Compared with field observation and laboratory experiment numerical simulation is less cost and more flexible in prescribing atmospheric parameters and emission conditions. Since last century numerical simulation of atmospheric environment in urban residence area is developed quickly. However the numerical simulation of atmospheric flow in urban residence area has some difficulties since it is of small scale motion in both spatial and temporal evolution. In environment engineering the analytic model is used.

Essentially pollutant dispersion is the particle motion carried by turbulent flows. The numerical simulation of pollutant dispersion must solve fluid flow equation and transportation equation of pollutant simultaneously. Modern computational fluid dynamics (CFD) is capable of predicting complex turbulent flows and pollutant dispersion numerically by means of turbulence modeling. There are two methods available in modeling turbulent flows, namely RANS (Reynolds average Navier-Stokes) and LES (Large Eddy simulation). The difference between RANS and LES lies in whether the turbulent fluctuations are partly resolved or not. RANS solves average Navier-Stokes equation and interaction of turbulent fluctuations on average motion is closed by casting models (RANS model). LES solves filtered Navier-Stokes equation, i.e. large scale turbulent motion, and the interaction of small scale turbulence on large scale motion is closed by LES model. LES has finer resolution than that of RANS; hence it costs more computation time; however LES can give much flow information, in particular it suitable for non-stationary turbulent flows (unsteady mean flows).

Environmental flows and pollutant dispersion in urban area are non-stationary phenomena due to the complex underlying surfaces and building layout. RANS can not correctly predict such unsteady turbulent flow while LES is a practical tool in numerical simulation of environmental flows in urban area. Hanna et al (2002) applied LES to study wind and turbulent field in a cubic building group. They showed that stronger street canyon effect arises in regular distribution of cubes rather than staggered displacement. Kanda et al (2004) used LES to investigate influence of building density on the turbulence propertied inside and outside of the canopy. Tseng et al. (2006) conducted numerical simulation of environmental flows in a real building group inside city Baltimore by LES and they showed complex building structure greatly influences the dispersion of hazard gas. The present authors (Shi et al.

Sixth International Symposium on Turbulence and Shear Flow Phenomena
Seoul, Korea, 22-24 June 2009

2008) investigate wind field and pollutant dispersion in a group of cubic buildings by LES and compared the numerical simulation with wind tunnel measurements (Davidson et al. 1996). The results show that LES is much better than RANS when subgrid stress model and boundary conditions are properly prescribed, for instance the Lagrangian dynamic model and turbulent fluctuations at inlet boundary.

In this paper LES is accepted to investigate the dispersion of traffic pollution in downtown Macao where is a heavy traffic and crowded populated area. The paper is organized as follows.

NUMERICAL METHOD

The governing equations of LES for both wind and concentration field can be written as follow

$$\partial \overline{u}_i / \partial x_i = 0 \tag{1}$$

$$\partial \overline{u}_i / \partial t + \overline{u}_j \partial \overline{u}_i / \partial x_j = -\partial \overline{p} / \rho \partial x_i + \nu \partial^2 \overline{u}_i / \partial x_j \partial x_j + \partial \tau_{ij} / \partial x_j + \overline{f}_i \tag{2}$$

$$\partial \overline{c} / \partial t + \overline{u}_j \partial \overline{c} / \partial x_j = D \partial^2 \overline{c} / \partial x_j \partial x_j + \partial T_j / \partial x_j + \overline{S}_c \tag{3}$$

in which ν and D are molecular viscosity and diffusivity respectively, $\overline{f}_i$ is an external force and $\overline{S}_c$ stands for the emission rate of contaminant, such as the exhaust from vehicles. In present computation the subgrid stress τ_{ij} and subgrid scalar flux T_j are closed by eddy viscosity and eddy diffusivity model as follows.

$$\tau_{ij} = 2\nu_t \overline{S}_{ij} + \tau_{kk}\delta_{ij}/3, \ \ \nu_t = C\Delta^2 \left|\overline{S}\right| \tag{4}$$

$$T_j = \kappa_t \partial \overline{c} / \partial x_j \ , \ \kappa_t = D\Delta^2 \left|\overline{S}\right| \ \text{or} \ \kappa_t = \nu_t / Pr_t \tag{5}$$

In equations (4) and (5) ν_t and κ_t are eddy viscosity and diffusion coefficients respectively. In this paper the Lagrange dynamic model, proposed by Meveneau et al. (1996), is used to determine coefficients C and D.

The finite volume method (FVM) is a well known discretization method with good conservative property in computational fluid dynamics. An improvement of numerical accuracy is utilized in this paper where the fourth order Padé type compact scheme is used for interpolation of surface average from volume average among element volumes. Non-staggered grids and fourth order Runge-Kutta integration in time advancement are used in computation. The details of the numerical formulation can be found in Xu et al. (2006).

Particular difficulty is to properly resolve the complicated building layout in numerical computation of micro-atmospheric flows in the residential areas. In this paper we use a composite model that the buildings and streets are properly resolved with fine grids in the residential area where the wind field and traffic pollution are most concerned while roughness element model is used for buildings in the surrounding area with coarse grids. In the properly resolved area the non-slip condition is satisfied at building surfaces and ground by Immersed Boundary Method (IBM). The composed model is a practical solution for satisfactory resolution for the complex canopy of city residential area with reasonable computation cost. The idea of IBM is to insert an additional force in the momentum equation at the grid points adjacent to the body surface that

$$\begin{aligned}\partial \overline{u}_i / \partial t + \overline{u}_j \partial \overline{u}_i / \partial x_j &= -\partial \overline{p} / \rho \partial x_i \\ &+ \nu \partial^2 \overline{u}_i / \partial x_j \partial x_j + \partial \tau_{ij} / \partial x_j + \overline{f}_i + g_i\end{aligned} \tag{6}$$

In Equation (6) g_i is non-zero only at the grid nodes near the solid boundary. There are various ways for prescribing the additional force to satisfy non-slip condition at solid surface, among others the most easy way is the direct force method (Mohd-Yusof, 1997) that

$$g_i^n = -RHS^n + \left(V_{bi} - u_i^n\right) / \Delta t \tag{7}$$

where the superscript n stands for the time step and term RHS contains advection, pressure gradient, diffusion and external force terms in Equation (6) and V_{bi} is the velocity at the body surface. In time advancement the velocity at the (n+1)th step is computed as

$$u_i^{n+1} - u_i^n = \Delta t\left(RHS^n + g_i^n\right) \tag{8}$$

The advantage of IBM is that the governing equation of fluid flows is computed numerically in Cartesian coordinates no matter how complicated the geometry of flow boundary is.

The roughness element model used in the paper is proposed by Belcher et. al. (2003) that a roughness drag force is inserted in the momentum equation that

$$f_i = -\left|U\right| U_i / L_c \tag{9}$$

The drag length scale L_c depends on the layout of the buildings in computational domain and its formula can be found in Belcher et al. (2003)

The computational domain is shown in Figure 1 which is a part of City Macao with horizontal extent 1200m×1200m and height of 400m. Area 1 is the most crowded residence area with heavy traffic and it is the major interesting place for numerical simulation of the traffic pollution. Mark B in area 1 is the location of field measurement of exhaust gas concentration. Mark A in area 3 is the location of field measurement of wind speed. In Area 1 the non-slip condition is used at ground and buildings by IBM while outside of Area 1 the roughness element model is used for buildings. The total grids are 157×178×76 for streamwise, spanwise and vertical directions respectively.

Wind is from left to right and the inlet boundary condition is prescribed by mean velocity with power law plus fluctuations as follows

$$U/U_h = \left(Z/Z_h\right)^{0.2} \tag{11}$$

in which U_h=2.23m/s and Z_h= 400m. The wind condition is corresponding to northwest wind on the 6th of September 2005. The velocity fluctuations are prescribed on inlet boundary that

$$u_i'(t) = u_i'(t-\Delta t)\cdot \exp\left(-t/T_i\right) + u_{igauss}' \tag{12}$$

in which u'_{igauss} is random fluctuations with Gaussian probability density and variance σ_u=0.4U_h, σ_v=0.3U_h, σ_w=0.1U_h, and time relaxation T_i=L_{ii}/σ_i with L_{ii}=60mm in the first term of equation (12) . The formula (12) was proposed by Hanna et. al. (2002) and we used it in the previous numerical simulation of plume dispersion in a building array in good agreement with experimental results (Shi et. al. 2008)

RESULTS

The numerical prediction of wind field at the cross road, i.e. location B in area 1 of Figure 1, are compared with wind tunnel measurements by Liu et al. (2000) shown in Figure 2. The numerical computation is in good agreement with wind tunnel measurements. In Figure 2 (b) a computation without inlet velocity fluctuations is also presented for comparison that the inlet velocity fluctuations is significant for correct prediction of turbulence intensity which is of great important for pollutant dispersion.

The computed wind speed and direction at location A of area 2 in Figure 1 are shown in Figure 3 in comparison with the field measurement.

The emission of CO from traffic is modelled as a surface source on the street at the height of 1m from the ground. The annual average of traffic emission coefficient in day time is shown in Figure 4 with a peak value of 3.511mg/m^2·s from field measurements.

Figure 5 shows the computed concentration of CO at the location of field measurements, i.e. B in Figure 1, and compare with monitoring data on the 6th of September 2005. The overall agreement is fairly good, however the peak value in the morning time is one hour delayed. The reason may be due to that we use annual average data of traffic emission which is the not the exact data in September 2005.

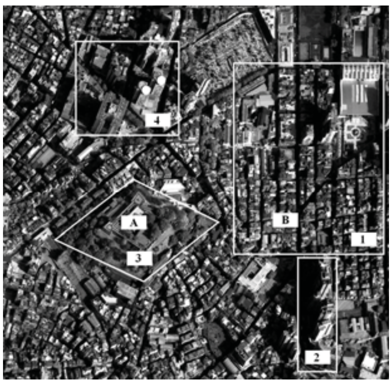
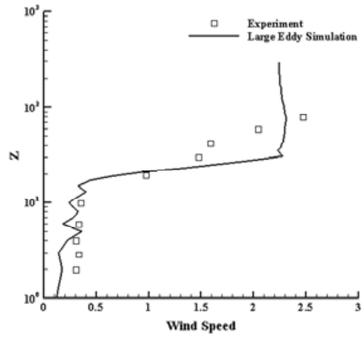
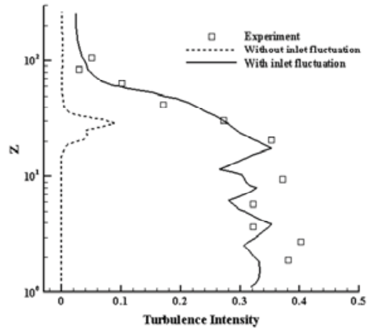


Figure 1 Aerial photograph of part of Macao

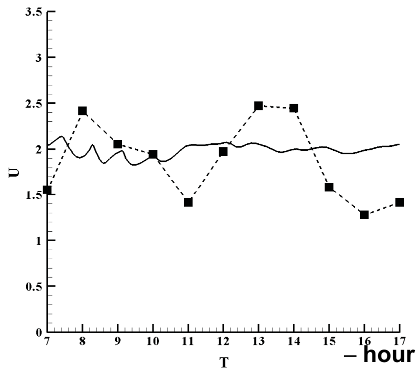


(a) Mean streamwise velocity profile

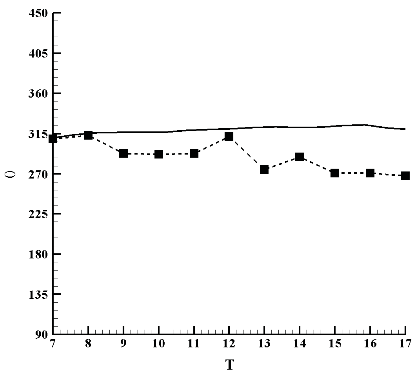


(b) Turbulence intensity profile

Figure 2 Wind speed and turbulence intensity at a cross road



(a) wind speed



(b) wind direction

Figure 3 The hourly variation of wind speed (m/s) and direction (degree) at location A in Figure 1.

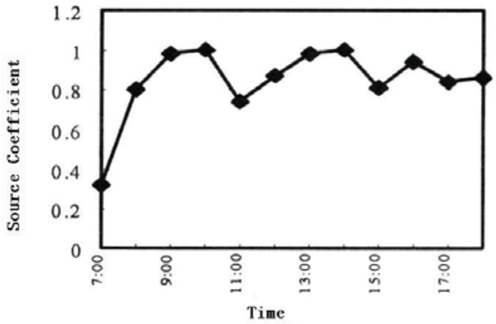


Figure 4 The source coefficient of traffic emission of CO

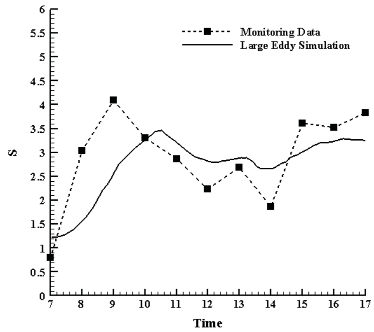


Figure 5 The hourly variation of concentration CO

CONCLUDING REMARKS

The above results indicate that Large Eddy Simulation is an appropriate tool for predicting micro atmospheric flows and contaminant dispersion. For such a complicated turbulent flow successful numerical simulation by LES is relied on the proper incoming condition, rational SGS model, and sufficient resolution etc. More results of wind field and pollutant dispersion in residence area will be given in final manuscript with detailed analysis.

ACKNOLEDGEMENTS

The authors would like to thank National Natural Science Foundation of China (NSFC Grant10872109, 10878204) and the Foundation for Development of Science and Technology in Macao (FDCT 055/2005/A).

REFERENCES

Belcher S. E. et al.. 2003 Adjustment of a turbulent boundary layer to a canopy of roughness elements. **Journal of Fluid Mechanics 488**: 369.

Davidson M.J. et al. 1996 “Wind tunnel simulation of plume dispersion through groups of obstacles“ Atmospheric Environment. **30**(22):3715-3731

Hanna S.R. et al. 2002 “Comparisons of model simulations with observations of mean flow and turbulence within simple obstacle arrays“ Atmospheric Environment 36: 5067-5079.

Kanda M, Moriwaki R, Kasamatsu F. 2004. Large-eddy simulation of turbulent organized structures within and above explicitly resolved cube arrays. **Boundary-Layer Meteorology 112**: 343-368

Liu B.Z. et al. 2000 Experimental study on the dispersion of vehicle exhaust in Macao. **Jouranl of Enviroment Science 20**:27 (in Chinese)

Lilly D.K. 1992. “A proposed modification of the Germano subgrid-scale closure method” Physics of Fluids**,** A4:633

Mohd-Yusof J. 1997 Combined Immersed Boundaries/B-Splines Methods for Simulations of Flows in Complex Geometries. **CTR Annual Research Briefs NASA Ames/Stanford University**, 317-328

Meneveau C. et al. 1996. “A Lagrangian dynamic subgrid-scale model of turbulence“ JFM, 319:385

Shi R.F. et al. 2008 Large eddy simulation of wind field and plume dispersion in building array. **Atmospheric Environment 42**:1083

Tseng Y H et al. 2006 Modeling flow around bluff bodies and predicting urban dispersion using large eddy simulation. **Environmental Science and Technology 40**: 2653- 2662

Xu L. et al. 2006 High accurate finite volume method for large eddy simulation of complex turbulent flows.**International Journal of Turbo and Jet-Engine 23**(3):191