Available online at www.sciencedirect.com
SCIENCE DIRECT®

ELSEVIER

Environmental Modelling & Software 20 (2005) 273–277

Environmental Modelling & Software

www.elsevier.com/locate/envsoft

Short communication

Study of pollution dispersion in urban areas using Computational Fluid Dynamics (CFD) and Geographic Information System (GIS)

A.K.M. Chu[a], R.C.W. Kwok[b], K.N. Yu[a,*]

[a]Department of Physics and Materials Science, City University of Hong Kong, Tat Chee Avenue, Kowloon Tong, Hong Kong
[b]Department of Public and Social Administration, City University of Hong Kong, Tat Chee Avenue, Kowloon Tong, Hong Kong

Received 12 December 2003; received in revised form 11 May 2004; accepted 26 May 2004

Abstract

The computational fluid dynamics software, CFX5.5 is employed to determine dispersions of emissions from vehicles traversing the streets. Information on the layouts and heights of buildings in the selected area is contained in a digitized map layer of buildings. The Geographic Information System software, ArcView 3.2a, with its programming facility, Avenue, has been used to extract the coordinates and heights of each building polygon under research. These are then input into the CFX-Build component of CFX5.5 to construct the geometry for simulations. The dispersion characteristics, such as the spread of the pollution dispersions, have been determined for different wind speeds and wind directions.

Keywords: Street canyons; Computational Fluid Dynamics; CFD; CFX; Geographic Information System; GIS

1. Introduction

The restricted geometry of city spaces with dense high-rise buildings will cause a complex flow, which can give rise to uneven distribution of pollutants. Many studies have been carried out on the flow regimes by field measurements, physical modelings and numerical simulations (DePaul and Sheih, 1985; Nakamura and Oke, 1988; Lee and Park, 1994; Meroney et al., 1996; Leitl and Meroney, 1997). Most previous studies have only considered the complex physical processes occurring within a single canyon or have analyzed these through two-dimensional vertical cross-sections. More recently, studies have been extended to more buildings or three-dimensional analyses have been carried out. These provide more realistic information on the dispersion of pollutants over an urban area.

The present paper proposes a methodology for such realistic calculations. Real layout of buildings and real dimensions of building structures are obtained through digitized maps and a Geographic Information System (GIS). The Computational Fluid Dynamics (CFD) technique will then be employed to study the dispersion of pollutants from the realistic urban area. For illustration purposes, a mixed residential and commercial area in Hong Kong will be chosen for our studies on the three-dimensional characteristics of pollutant dispersion for various wind directions and wind speeds.

2. Model validation

The developed models have to be verified before they can be applied to real life simulations. To test the performance of the present wind field model, simulations

---

* Corresponding author. Tel.: +852 2788 7812; fax: +852 2788 7830.
E-mail address: peter.yu@cityu.edu.hk (K.N. Yu).

1364-8152/$ - see front matter
doi:10.1016/j.envsoft.2004.05.007

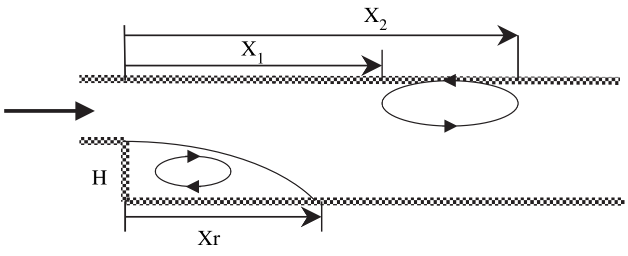

Fig. 1. Schematic diagram of a backward-facing step. Xr is the reattachment length; X_1 and X_2 define the beginning and the end of the recirculation region, respectively, at the upper wall; H is the height of the step.

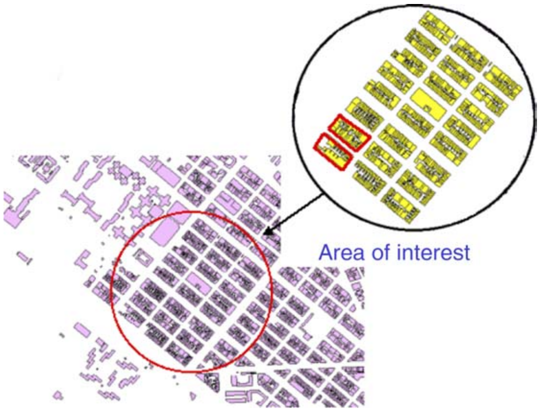

Fig. 3. A selected area of interest within the Sham Shui Po district chosen for our detailed studies on the realistic dispersion characteristics. To simplify the simulations, buildings which are close to each other in the same block are grouped together and regarded as a single building (shown as red rectangles).

have been performed with flow parameters and geometry of the test section the same as those of the experiment carried out by Armaly et al. (1983). Fig. 1 shows the schematic diagram of the backward-facing step used in the test section. The backward-facing step is regarded as a standard test for evaluating the stability and accuracy of numerical simulations for incompressible flow problems.

Numerical simulations of the test section were performed with different Reynolds numbers between 70 and 7500 to compute the reattachment lengths and the streamline patterns for comparison with those from the experiments. The range of the Reynolds numbers covers the laminar, transitional and part of the turbulent regimes of the flow. The numerical and experimental results for the characteristic lengths, i.e., reattachment and detachment lengths, are shown in Fig. 2 as a function of Reynolds number. It can be seen from Fig. 2 that the simulation results agree very satisfactorily with the experimental results for different regimes, i.e., laminar ($Re < 1200$), transitional ($1200 < Re < 6600$) and turbulent ($Re > 6600$) regimes. The locations of the primary circulation and secondary circulation regions can be deduced from Fig. 2. For most of the investigated range of the Reynolds numbers, the beginning and end of the recirculation region at the upper wall are, respectively, upstream and downstream of the reattachment point of the primary circulation region.

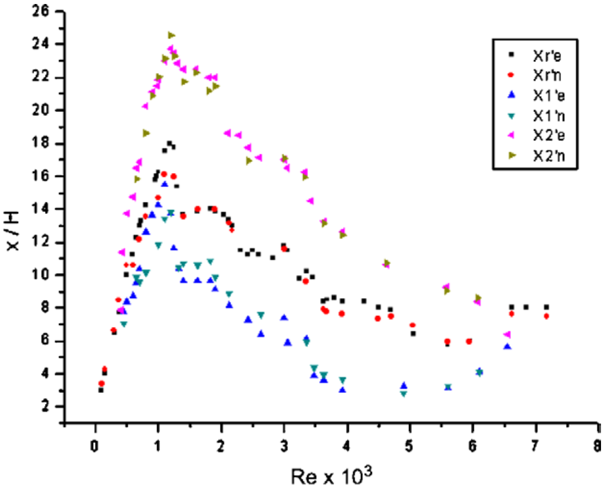

Fig. 2. Comparisons between the experimental and numerical results for the characteristic lengths with different Reynolds numbers; with e denoting experiment results and n denoting numerical results. Other symbols are defined as those in Fig. 1.

3. Street canyon simulations

3.1. Computational techniques and data

3.1.1. Geographic Information System (GIS)

The software ArcView GIS 3.2a (ESRI) is used in this study. Realistic information on the layouts and heights of building is contained in a digitized map layer of buildings in Hong Kong. GIS with its programming facility, Avenue, has been used to extract the coordinates and heights of each building polygon under research, and that information is exported to a Microsoft Excel file.

3.1.2. CFX

The CFD code CFX5.5 (AEA Technology, Harwell) is used for simulating the wind flow and pollutant

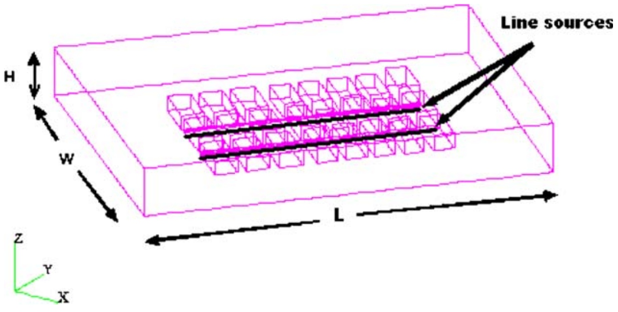

Fig. 4. Schematic diagram of the model. The two parallel-line sources represent emissions from vehicles traversing the main streets.

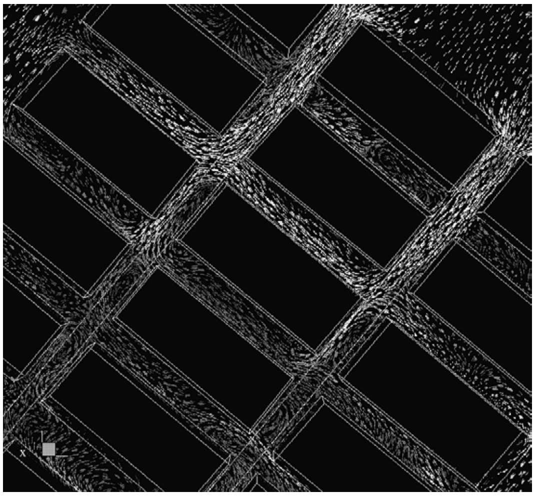

Fig. 5. Computed wind field around the buildings.

concentration patterns over an urban area. Those coordinates and the heights of buildings obtained from GIS are then input into the pre-processor of CFX5.5 to construct the geometry for the simulations.

3.1.3. Monitoring area

In the present study, as an example, a mixed residential and commercial area, Sham Shui Po, is chosen as our studied area. The map of a selected area in Sham Shui Po is shown in Fig. 3. A special feature of this district is attributed to its proximity to the old Kai Tak International Airport of Hong Kong, so that the height restriction has given rise to a regular array of streets with buildings all of very similar heights. In order to simplify the simulations, buildings which are close to each other and which have similar heights in the same block are grouped together and regarded as a single building. Fig. 3 has shown two such examples by the red rectangles.

3.2. Simulation model

Fig. 4 shows the schematic diagram of the simulation model. The software CFX5.5, coupled with a k–ϵ turbulence model, is employed to determine the dispersion of pollutants from the two parallel-line pollutant sources, which represent emissions from vehicles traversing the main streets.

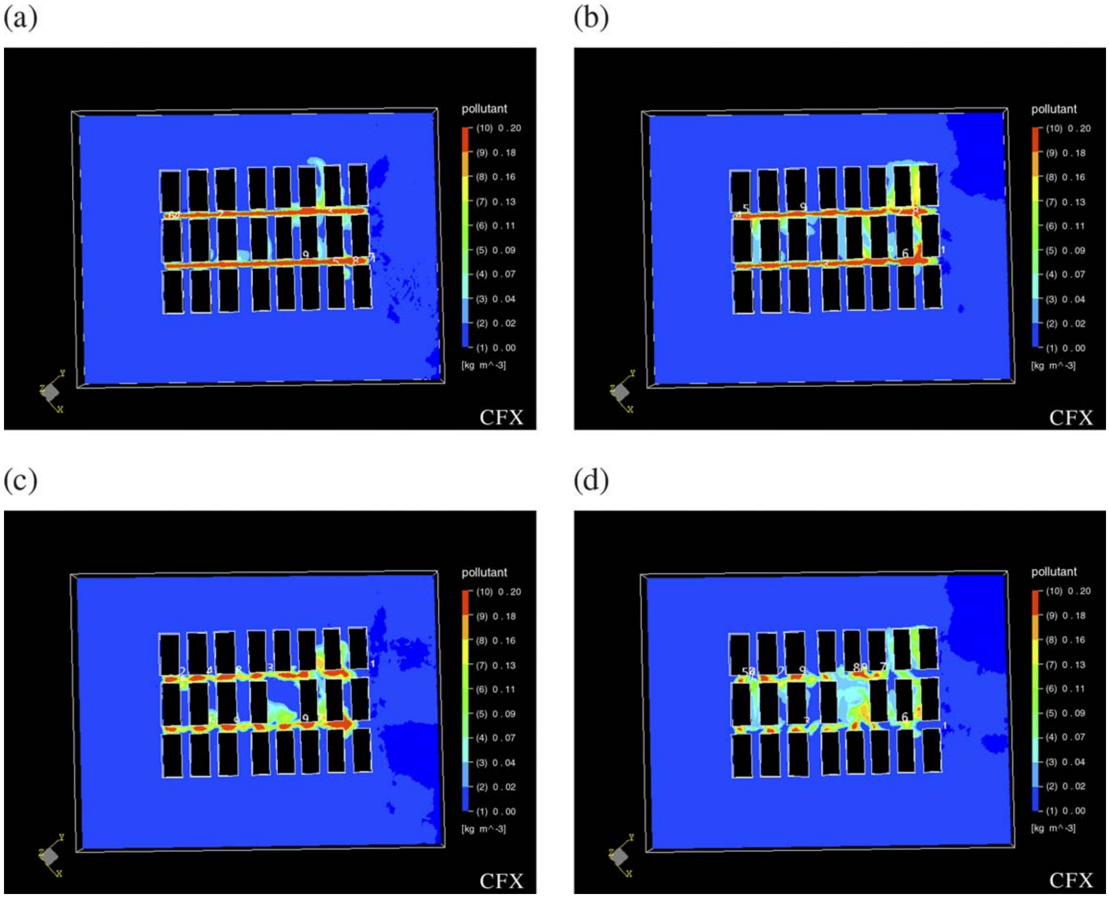

Fig. 6. Plan-view of the selected area showing the characteristics of the pollutant dispersion: (a) for low elevation at 2 m, and for easterly wind with wind speed $= 1.5$ m s^{-1}; (b) for low elevation at 2 m, and for easterly wind with wind speed $= 5$ m s^{-1}; (c) for high elevation at 20 m, and for easterly wind with wind speed $= 1.5$ m s^{-1}; (d) for high elevation at 20 m, and for easterly wind with wind speed $= 5$ m s^{-1}.

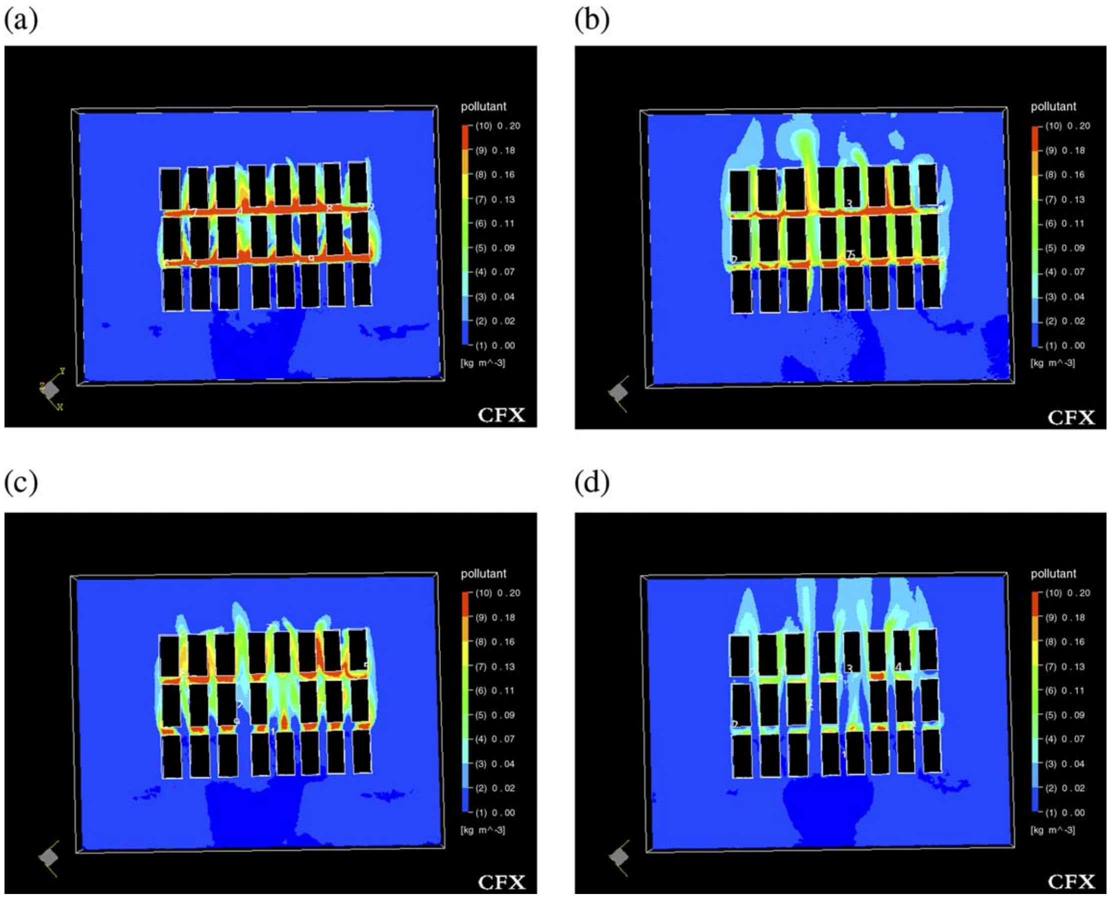

Fig. 7. Plan-view of the selected area showing the characteristics of the pollutant dispersion: (a) for low elevation at 2 m, and for southerly wind with wind speed = 1.5 m s^{-1}; (b) for low elevation at 2 m, and for southerly wind with wind speed = 5 m s^{-1}; (c) for high elevation at 20 m, and for southerly wind with wind speed = 1.5 m s^{-1}; (d) for high elevation at 20 m, and for southerly wind with wind speed = 5 m s^{-1}.

Simulations for four wind directions (two parallel and two perpendicular to the line sources) and two wind speeds (1.5 and 5 m s^{-1}) are considered in the present study.

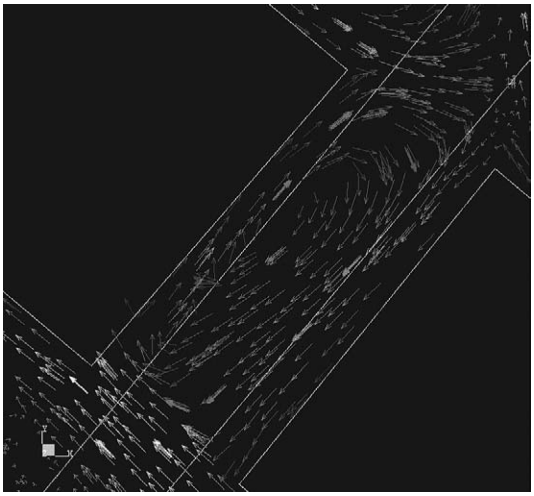

Fig. 8. A clockwise vortex inside the canyons between two junctions.

These two wind speeds are chosen because they represent the typical low and high wind speeds in the area.

4. Results and discussion

A very complex wind pattern has been found around the buildings, which is shown in Fig. 5. The wind velocities have been found to be large at the edges of the buildings near the inlet and decrease in weak wake areas. The wind velocities are reduced dramatically inside the canyons between two junctions and stagnation areas are formed there. This kind of uneven distribution of wind velocities will definitely influence the pollutant dispersion.

In our results, the pollutant distributions of easterly and southerly winds are similar to those of westerly and northerly winds, respectively. As such, we will only discuss the cases for easterly and southerly winds. The plan-view cross-sections of the canyons have been shown for easterly and southerly winds in Figs. 6 and 7, respectively.

4.1. Easterly wind (parallel to line sources)

Fig. 6a shows the pollutant distribution for the wind speed of 1.5 $\mathrm{m\,s^{-1}}$ at 2 m elevation. The pollutants are confined within the canyons. At such a low elevation, and with a small wind speed, the buildings are blocking the dispersion of the pollutants. If the wind speed increases to 5 $\mathrm{m\,s^{-1}}$, as shown in Fig. 6b, the pollutant at 2 m elevation has started to penetrate into the perpendicular canyons close to the inlet.

Different conditions occur for the cases of high elevation at 20 m above ground. The different heights of buildings, which reflect the realistic case and which have been implemented in our simulation model, would also affect the distribution of pollutants. Fig. 6c and d show that the concentrations of pollutants are relatively lower than those at the 2 m level because time is needed for the pollutants to reach higher levels. Furthermore, due to turbulence between the buildings, the pollutants spread much far away when compared to those at the 2 m level.

4.2. Southerly wind (perpendicular to line sources)

At the low elevation of 2 m with a smaller wind speed of 1.5 $\mathrm{m\,s^{-1}}$, as shown in Fig. 7a, the pollutants are confined to the selected area. For a larger wind speed of 5 $\mathrm{m\,s^{-1}}$, as shown in Fig. 7b, the pollutants have started to leak out from this selected area.

At the high elevation of 20 m, as shown in Fig. 7c and d, the pollutant concentrations have much larger spreads and have become unconfined, even under the low wind-speed condition. However, relatively high pollutant concentrations are usually left in the canyons between successive junctions. A closer look at the wind fields reveals that the detainment of the high pollutant concentrations is due to the existence of vortices between successive junctions, so that the pollutants cannot be easily dispersed away. Fig. 8 shows an enlarged view of the airflow and there is a clockwise vortex between two junctions. Besides, when the wind becomes stronger, as shown in Fig. 7d, it is easy to notice the very low pollution concentrations around the buildings.

4.3. Special feature of the simulated geometry

There is a special feature in the selected area: a building whose height is smaller than 20 m and forms a well-shape feature. From Figs. 6c and d, and 7c and d, we can see that this well-shape feature has helped the dispersion of the pollutants.

The pollutant dispersion patterns for parallel and perpendicular wind flows with wind speed of 5 $\mathrm{m\,s^{-1}}$ observed at the 20 m elevation shown in Figs. 6d and 7d, respectively, are compared. For parallel wind flows, lowering of the pollutant concentration in the area of the well-shaped feature is lowered, although not very significantly. On the contrary, for perpendicular wind flows, a significant decrease of the pollutant concentration has been observed.

References

Armaly, B.F., Durst, F., Pereira, J.C.F., Schonung, B., 1983. Experimental and theoretical investigation of backward-facing step flow. Journal of Fluid Mechanics 127, 473–496.

DePaul, F.T., Sheih, C.M., 1985. A tracer study of dispersion in an urban street canyon. Atmospheric Environment 19, 555–559.

Lee, I.Y., Park, H.M., 1994. Parameterization of the pollutant transport and dispersion in urban street canyons. Atmospheric Environment 28, 2343–2349.

Leitl, B.M., Meroney, R.N., 1997. Car exhaust dispersion in a street canyon. Numerical critique of a wind tunnel experiment. Journal of Wind Engineering and Industrial Aerodynamics 67/68, 293–304.

Meroney, R.N., Leitl, B.M., Rafailidis, S., Schatzmann, M., 1996. Study of line source characteristics for 2-D physical modeling of pollutant dispersion in street canyons. Journal of Wind Engineering and Industrial Aerodynamics 62, 37–56.

Nakamura, Y., Oke, T.R., 1988. Wind, temperature and stability conditions in an east–west oriented urban canyon. Atmospheric Environment 22, 2691–2700.