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

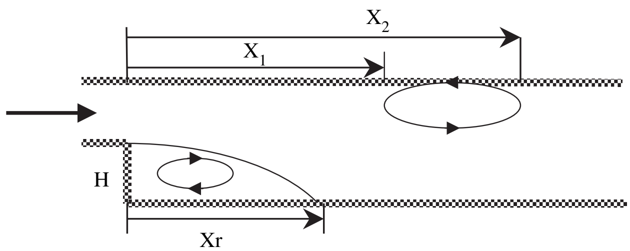

Fig. 1. Schematic diagram of a backward-facing step. Xr is the reattachment length; X_1 and X_2 define the beginning and the end of the recirculation region, respectively, at the upper wall; H is the height of the step.

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

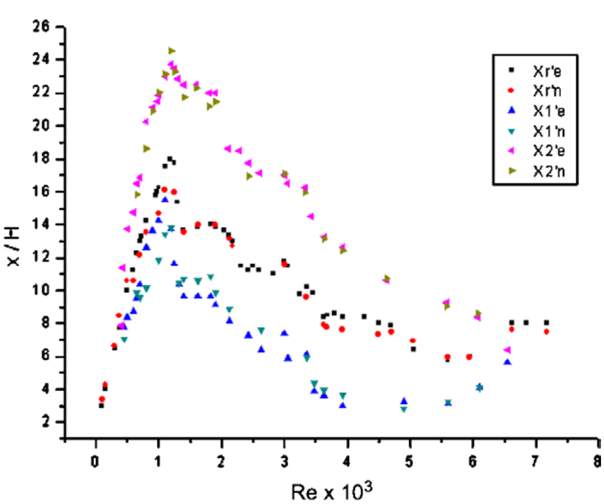

Fig. 2. Comparisons between the experimental and numerical results for the characteristic lengths with different Reynolds numbers; with e denoting experiment results and n denoting numerical results. Other symbols are defined as those in Fig. 1.

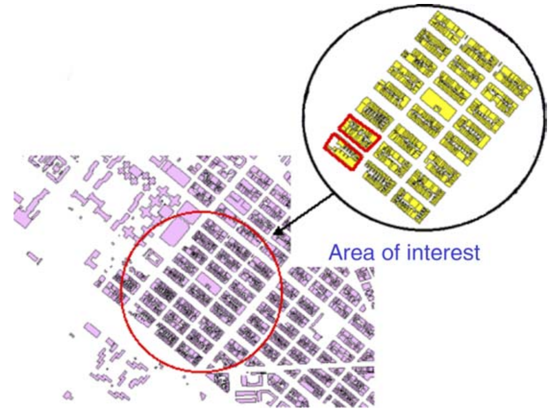

Fig. 3. A selected area of interest within the Sham Shui Po district chosen for our detailed studies on the realistic dispersion characteristics. To simplify the simulations, buildings which are close to each other in the same block are grouped together and regarded as a single building (shown as red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

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.

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

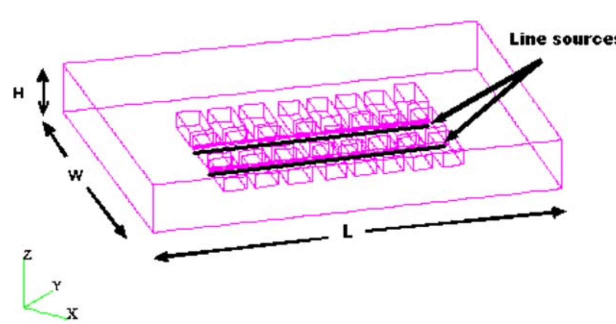

Fig. 4. Schematic diagram of the model. The two parallel-line sources represent emissions from vehicles traversing the main streets.