H13-48b

FLOW AND POLLUTANT DISPERSION WITHIN THE CANAL GRANDE CHANNEL IN VENICE (ITALY) VIA CFD TECHNIQUES

Riccardo Buccolieri[1,2], Flavio Sartoretto[1], Achille Giacometti[2], Silvana Di Sabatino[2], Laura Sandra Leo[2], Beatrice Pulvirenti[4], Mats Sandberg[5], Hans Wigö[5]

[1]Dipartimento di Informatica, Università "Ca' Foscari" Venezia, Via Torino 155, 30172 Mestre-Venezia, Italy
[2]Dipartimento di Scienza dei Materiali, University of Salento, Via Monteroni, 73100 Lecce, Italy
[3]Dipartimento di Chimica Fisica, Università "Ca' Foscari" Venezia, Dorsoduro 2137, 30121 Venezia
[4]Dipartimento di Ingegneria Energetica, Nucleare e del Controllo Ambientale, University of Bologna, Bologna, Italy
[5]Division of Indoor Environment, Department of Technology and Built Environment, KTH Research School, University of Gävle, SE-80176 Gävle, Sweden

Abstract: The present paper is aimed at the analysis of flow and pollutant dispersion in a portion of the Canal Grande (Grand Canal) in Venice (Italy) by means of both Computational Fluid Dynamics (CFD) FLUENT simulations and wind tunnel experiments performed at the University of Gävle (Sweden). For this application, Canal Grande can be viewed as a sort of street canyon where the bottom surface is water and bus boat emissions are the major source of pollution. Numerical investigations were made to assess the effect of the water surface on air flow and pollutant concentrations in the atmosphere. One of the challenges has been to deal with the interface between two immiscible fluids which requires ad-hoc treatment of the wall in terms of the numerical scheme adopted and the grid definition which needs to be much finer than in typical numerical airflow simulations in urban street canyons. Preliminary results have shown that the presence of water at the bottom of the street canyon modifies airflow and turbulence structure with direct consequences on concentration distribution within the domain.

Key words: *street canyon, Canal Grande, CFD simulations, multiphase model, wind tunnel experiments.*

INTRODUCTION

The city of Venice (Italy) lies in the middle of a 550 km^2-wide lagoon (close to the Adriatic Sea) and is mainly fed by fresh water from a number of rivers originating from the main land. It consists of over one hundred small islands, separated by a large number of channels that communicate with the lagoon. The main channel of this complex network is the famous Canal Grande (Grand Canal). Other channels spread out in all directions until reaching the lagoon. The city of Venice has been the subject of many studies, most of them focusing on water ecological problems and hydrodynamic behaviours of the channels. However, urban air quality, urban heat island and ventilation problems are becoming increasingly important especially in connection with the need of preserving the prestigious historical and monumental heritage. More recently, air quality investigations have been carried out mainly by Venice University, Public Administration and the Veneto Region Environmental Protection Agency (ARPAV) even though pollutant concentration in city channels have not been specifically studied. Motivated by the requirement of evaluating air quality in Venice (Italy), we first addressed the question whether traditional air quality models were suitable for this special urban environment. Recent literature has documented an increasing development of suitable atmospheric dispersion models for urban air quality management (Vardoulakis, S. *et al.*, 2003; Holmes, N. and L. Morawska, 2006). At the micro-scale, full Computational Fluid Dynamics (CFD) models have become increasingly popular and attractive because they allow to evaluate directly the effects of complex building arrangements and shapes, though they are still computationally expensive for routine. It was important in the context of this study to evaluate whether classical CFD modelling could be applied to Venice where most street canyons are represented by the city channels.

In this framework the present paper deals with the analysis of flow and pollutant dispersion in street canyons where water constitutes the bottom domain of the street canyon by means of CFD simulations. Some preliminary wind tunnel tests have also been performed to complement numerical investigations. At first, we investigated flow and pollutant dispersion in a simplified street canyon with aspect ratio height/width (H/W) ~ 0.3. The aim is to study flow behaviour of both air and water and compare it with flow regimes and pollutant patterns that develop in typical solid-surface urban street canyons. This case is a simplification of the real scenario considered in a second step, where we analyse flow and pollutant dispersion within a portion of the Canal Grande. The starting point for the CFD study is the methodology generally applied for the study of dispersion in typical urban areas. This study constitutes a first step towards a novel CFD research application.

DESCRIPTION OF THE CASES INVESTIGATED

Canal Grande is a channel which crosses the city of Venice from North to South shaping out a big 'S' as shown in Figure 1a. It is about 4 km long, about 30 to 90 m wide with an average depth of about 5 m. The Canal Grande banks are aligned with slightly more than 170 buildings. It can be considered a sort of street canyon whose bottom surface is not solid but water (Figure 1b). Public transport is provided by water boats and private water taxis whose emissions are expected to be the major source of pollution.

For the purpose of the present work, a portion of the channel has been considered. The chosen portion is characterized by several buildings of different height, with the maximum height equal to about 26 m. The street width is equal to about 65 m, resulting in an aspect ratio H/W of about 0.3 (Figure 1c).

Figure 1. a) Top view of the Venice city showing the Canal Grande shaping out a big "S" (from Insula S.p.a.). b) Image of the Canal Grande (from http://www.italiaabc.it/monumenti/venezia/canal-grande.html). c) Portion of the Canal Grande considered in this study (from Insula S.p.a.).

First we consider a simplified geometry of the portion of the Canal Grande shown in Figure 1a. It consists of an idealized symmetric street canyon (model scale 1:200) of aspect ratio H/W ~ 0.3, H=10cm. In order to deduce how the presence of water at the bottom of the canyon (*water-street canyon* hence forth) can alter the flow regimes that are characteristic of typical urban *no water-street canyons* (Oke, T.R., 1987), CFD simulations have been performed for both cases. The wind direction is perpendicular to the street axis as shown in Figure 2a. Moreover, preliminary wind tunnel tests have been carried out as a first attempt to understand the flow regimes which develop in such a street system.

In order to further substantiate and extend the numerical and wind tunnel investigations within the above idealized street canyon, the next step has been to study the real scenario (model scale 1:200) of the Canal Grande by means of CFD simulations. For this analysis, as an example, the wind approaches from the left (East) and it is inclined of about 45° to the street axis (Figure 2b).

CFD MODELLING SETUP AND WIND TUNNEL TESTS

Idealized street canyon

<u>*No water-street canyon*</u>. 3D steady-state simulations were performed by means of the CFD code FLUENT (2006). Generally, main recommendations provided in the COST Action 732 (2005-2009) were implemented. The computational domain was discretized using approximately one million hexahedral elements. The smallest elements lengths were δxmin = 0.05H, δymin = 0.1H, δzmin = 0.025H within the street canyon volume. The distance from the inlet boundary to the first building of the street canyon was 8H, the distance of the lateral boundaries from the street canyon ends was 15H, the distance from the top of the domain to the building roof was 9H and the distance from the outflow boundary to the downstream building was 30H. The Reynolds Stress Model (Launder, B. E *et al.*, 1975) was used. The inlet wind speed *U(z)* was assumed to keep to the following power law profile :

$$\frac{U(z)}{U_{ref}} = \left(\frac{z}{z_{ref}} \right)^{0.35} \tag{1}$$

where U_{ref} = 8.13ms^{-1} is the undisturbed wind speed at z_{ref} = 0.75m (model scale). Turbulent kinetic energy and dissipation rate profiles were specified as follows:

$$k = \frac{u_*^{\,2}}{\sqrt{C_\mu}}\left(1 - \frac{z}{\delta}\right), \; \varepsilon = \frac{u_*^{\,3}}{\kappa z}\left(1 - \frac{z}{\delta}\right) \tag{2}$$

where u_* = 0.44ms^{-1} is the friction velocity, κ the von Kàrmàn constant (0.40), C_μ = 0.09 and δ the model scale boundary layer depth (1m). Symmetry boundary conditions were specified for the top and lateral sides of the computational domain. For the boundary downwind of the street canyon, a pressure-outlet condition was used.

For dispersion, the advection diffusion module was applied. A ground level source emitting N_20 was used to simulate the release of traffic exhausts, with an emission rate *Q* equal to 10g/s. Mean gas concentrations *c* are normalized as follows:

$$c^+ = \frac{c\, U_{ref}\, H}{Q/l} \tag{3}$$

where *l* is the line source length. The second order discretisation scheme for pressure and the second order upwinding numerical schemes for momentum, k, ε, Reynolds Stresses and the scalar were used for better accuracy and limited/reduced numerical diffusion. The SIMPLE scheme was selected for the pressure-velocity coupling.

<u>*Water-street canyon*</u>. 2D steady-state simulations were performed by applying the multiphase module available in FLUENT as a first attempt to determine the appropriate model set-up. Currently there are two approaches for the numerical calculation of multiphase flows: the Euler-Lagrange approach and the Euler-Euler approach. In the Euler-Euler approach, the different phases are treated mathematically as interpenetrating continua. Since the volume of one fluid (e.g. gas) in a specific phase cannot be occupied by the other fluid in a different phase (e.g. liquid), the concept of phasic volume fraction is introduced. These volume fractions are assumed to be continuous functions of space and time and their sum is equal to one. Conservation equations for each fluid in a specific phase are derived to obtain a set of equations, which have similar structure for all fluid phases. These equations are closed by providing constitutive relations that are obtained from empirical information. In particular, three Euler-Euler multiphase models are available: the volume of fluid (VOF) model, the mixture model and the Eulerian model (FLUENT, 2006)

For the purpose of the present study, which is a "free-surface flow" problem, the VOF model was used as the only model which accounts for immiscible fluids. With this approach we were able to model air and water by solving a single set of momentum equations and tracking the volume fraction of each of the fluids throughout the domain. The VOF formulation relies on the fact that those fluids (or phases) are not interpenetrating. In each control volume, the volume fractions of all phases sum to one. The fields for all variables and properties are shared by the phases and represent volume-averaged values, as long as the volume fraction of each of the phases is known at each location. Thus the variables and properties in any given cell are either purely representative of one of the phases, or representative of a mixture of the phases, depending upon the volume fraction values. The effects of surface tension (0.073 Nm^{-1}) along the water-air interface was also accounted for, introducing a source term in the momentum equation by applying the continuum surface force (CSF) model proposed by Brackbill, J. U. *et al.* (1992).

We used conservative numerical settings for typical VOF model by using the PRESTO discretisation scheme for pressure and the modified HRIC for volume fraction. The second order upwinding discretisation scheme was employed for momentum, k, ε and Reynolds Stresses in order to increase the accuracy and reduce numerical diffusion. A preliminary grid study was also performed starting from a similar grid size used in 3D simulations.

A similar *water-street canyon* case was also reproduced at the Department of Technology and Built Environment of the University of Gävle (Sweden) in a closed-circuit wind tunnel with a working section of 11m long, 3m wide and 1.5m high. Within this preliminary study only few videos have been stored. Several small plastic floating objects were employed to visualize the water movement at several approaching wind speeds and aspect ratios (~ 0.3, ~ 0.6 and ~ 0.9) and from the observation, the flow pattern could be determined.

Real scenario: the Canal Grande

In the second step, flow and pollutant dispersion within a portion of Canal Grande shown in Figure 1b were investigated by mean of CFD simulations. Main geometry details of the study area were taken from a digital CAD file available from Insula S.p.a. (Società per la Manutenzione Urbana di Venezia). As a preliminary test, and for the sake of computational convenience, the real geometry was slightly simplified still maintaining the main geometrical details of the street canyons and by using full scale cell dimensions $\delta xmin = \delta ymin = 2m$, $\delta zmin = 0.5m$ till a height of 5m since the emphasis was on pedestrian pollutant exposure. The number of the computational cells was approximately one million. 3D steady-state simulations were performed using a computational domain whose dimensions are similar to those used in the wind tunnel street canyon case, being H the full scale height of the tallest building (Hmax ~ 26m). Velocity, turbulent kinetic energy and dissipation rate profiles were specified at the inlet as already defined in equations (1) and (2). For preliminary testing, and due to convergence and numerical diffusion issues, only the *no water* case was simulated by applying the same set-up used in the idealized street canyon.

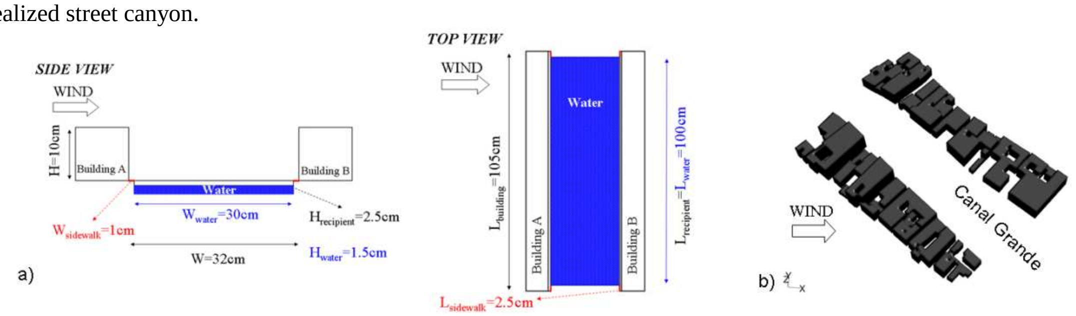

Figure 2. a) Geometric details of the idealized street canyon of aspect ratio H/W ~ 0.3 (model scale 1:200) used both in wind tunnel test and in CFD analysis. b) Sketch of the Canal Grande portion geometry considered in CFD simulations.

RESULTS

Idealized street canyon

Figure 3a shows velocity vectors of wind velocity magnitude (normalized by U_{ref}) and normalized concentration contours at two horizontal planes, i.e. at z = 0.1H (2m in full scale) and z = 0.5H (10m in full scale) obtained from CFD simulations. Overall, large pollutant concentrations in the middle part of both leeward wall A and windward wall B are found in the *no water-street canyon*. The decreasing of concentration level towards the street ends can be explained by the enhanced natural ventilation at those points. In particular, the figure highlights the presence of two symmetric corner vortices, one clockwise and the other anticlockwise, close to the ground in the *no water* case. In the middle of the street canyon, those vortices impinge on two smaller vortices rotating in the opposite direction shaping out two "8"-like symmetric vortices throughout the length of the canyon. This implies that air moves in the wind direction in the middle of the canyon leading to an accumulation of pollutants and larger concentration values at the windward wall rather than at the leeward wall.

Wind tunnel tests carried out for water-street canyon case have shown that the floating objects move through two symmetric vortices at all the wind speeds and aspect ratios considered. This means that, contrary to what happens in the *no water-street canyon* case, in the *water* case floating objects reaching the middle of the canyon move opposite to the wind direction as shown in Figure 3b. 3D simulations with water are therefore needed to further resolve the air-water behaviour at the interface.

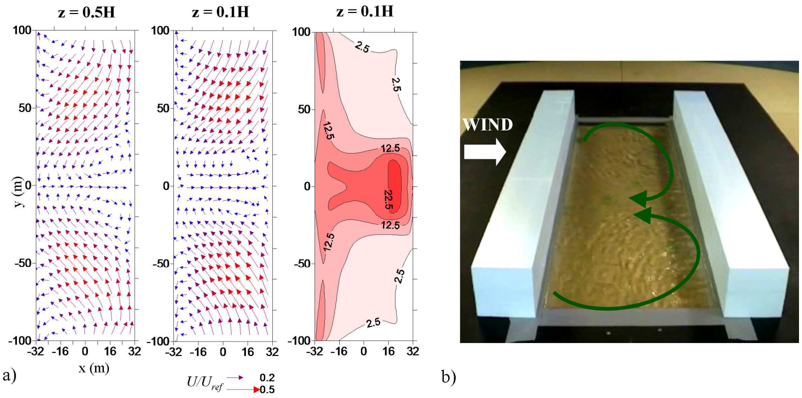

Figure 3. a) *No water-street canyon*. Vectors of normalized wind velocity obtained from CFD simulations. b) *Water-street canyon*. Wind tunnel frame showing water waves and movement of green floating objects following two symmetric corner vortices.

Although 3D simulations are required to further explore the above phenomenon as remarked, 2D simulations in the *water* case do show that the typical vertical vortex occurring for *no-water* canyon is partly modified as depicted in Figure 4. This suggests that the water body has a non-negligible influence on airflow particularly in the vicinity of the interface with the atmosphere. This results in a modification of the pollutant dispersion behaviour. Numerical simulations produced only small waves at the interface as shown in Figure 4b and some problems have been identified such as a large numerical diffusion and lack of good convergence of the continuity and volume fraction.

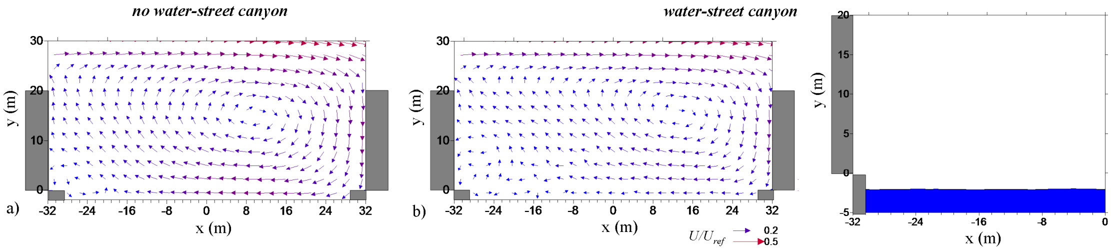

Figure 4. a) *No water-street canyon*. Vectors of normalized wind velocity. b) *Water street canyon*. Vectors of normalized wind velocity and water volume fractions showing waves at the interface obtained from 2D transient CFD simulations.

Further 2D and 3D simulations, both steady-state and unsteady, and a systematic grid study are in progress to enable appropriate set-up of CFD methodology. New systematic wind tunnel experiments have also been planned.

Real scenario: the Canal Grande

In this section we discuss CFD results of flow and pollutant dispersion within the *no water* scenario of Canal Grande for one approaching wind direction, i.e. East. Figure 5 shows vectors of the normalized velocity magnitude and concentration contours at z = 2m. Due to the interaction with the built environment, the resulting flow is channelled along the Canal Grande blowing from North to South. A helical flow develops inside the canyon yielding a forward convective transport of pollutants and larger concentrations are found at the leeward-oriented wall. Large concentration bubbles occur due to the formation of small vortices in correspondence of lateral street openings. In order to determine the effect on flow and pollutant dispersion due to the presence of water at the bottom of the Canal Grande, CFD simulations will be performed once the suitable set-up of CFD methodology will be achieved.

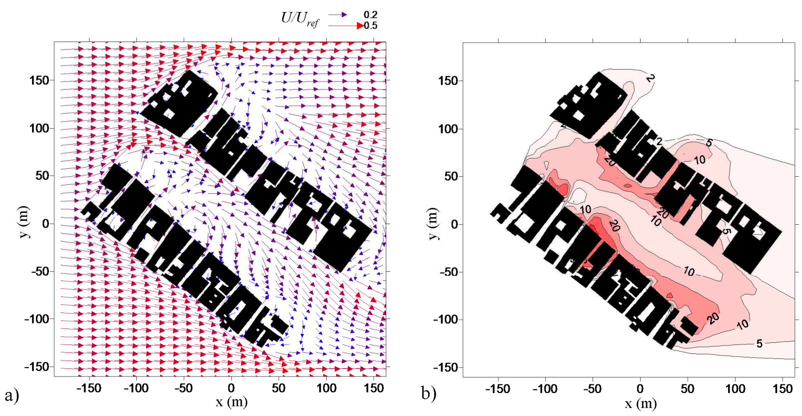

Figure 5. Real scenario. Vectors of normalized wind velocity magnitude (a) and contours of normalized concentration at z = 2m (full scale) obtained from CFD simulations.

CONCLUSIONS

The goal of the present work was the analysis of flow and pollutant dispersion in a portion of the Canal Grande in Venice (Italy) by means of CFD investigations and wind tunnel experiments. Preliminary numerical results employing the VOF module available in FLUENT and wind tunnel investigations were also performed in a simplified street canyon with the aim of finding the appropriate set-up when simulating flow in a street canyon where water makes up the bottom domain in lieu of solid grounds/roads.

A preliminary analysis of numerical results in the idealized *no water-street canyon* suggests that for a perpendicular approaching wind the atmospheric flow above the water should follow a horizontal 8-shaped vortex with air flowing in the same direction as the wind, in the middle of the canyon. On the other hand, experimental tests of the *water* case showed that the floating objects at the water-air interface follows two symmetric vortices moving opposite to the approaching wind direction in the middle of the canyon. Furthermore, the typical vertical vortex inside the canyon was modified due to the presence of water. To completely understand these phenomena, further testing is in progress since a number of problems have arisen in the steady-state CFD simulations VOF set-up, such as lack of convergence and large numerical diffusion.

ACKNOWLEDGEMENTS

The authors wish to thank “Insula S.p.a. - Società per la Manutenzione Urbana di Venezia” for providing the CAD geometry of Venice, Mr. Salim Mohamed Salim from University of Nottingham (Malaysia), Mr. Mirco Magnini and Mr. Alexandro Palmieri both from University of Bologna for useful discussions.

REFERENCES

Brackbill, J. U., D. B. Kothe and C. Zemach, 1992: A continuum method for modelling surface tension. *J. Comput. Phys.*, **100**, 335-354.

COST Action 732, 2005–2009: Quality assurance and improvement of micro-scale meteorological models. http://www.mi.uni-hamburg.de/Home.484.0.html

FLUENT, 2006: V6.3 User’s Manual. http://www.fluent.com

Holmes, N. and L. Morawska, 2006: A review of dispersion modelling and its application to the dispersion of particles: an overview of different dispersion models available. *Atmospheric Environment*, **40**, 5902-5928.

Oke, T. R., 1987: Boundary Layer Climates, 2nd Edition, Methuen, UK.

Vardoulakis, S., B. E. A. Fisher, K. Pericleous and N. Gonzalez-Flesca, 2003: Modelling air quality in street canyons: a review. *Atmospheric Environment*, **37**, 155-182.