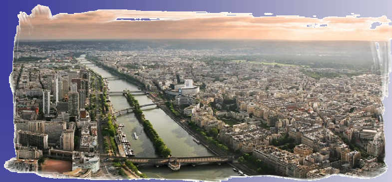


UNIVERSITY OF VENICE (ITALY)

UNIVERSITY OF SALENTO (ITALY)

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

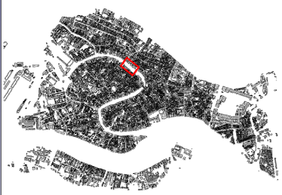
[4]Dipartimento di Ingegneria Energetica, Nucleare e del Controllo Ambientale, University of Bologna, Bologna, Italy

[5]Division of Indoor Environment, Department of Technology and Built Environment, KTH Research School, University of Gävle, SE-80176 Gävle, Sweden

HARMO13

13th International Conference on HARMONISATION WITHIN ATMOSPHERIC DISPERSION MODELLING FOR REGULATORY PURPOSES - Paris (France), 1-4 June 2010

OBJECTIVE: 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



- **Venice (Italy)** lies in the middle of a 550 km²-wide lagoon (close to the Adriatic Sea)
- 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**, from North to South shaping out a big 'S'.

CANAL GRANDE

http://it.wikipedia.org/wiki/Canal_Grande

- About 4 km long, about 30 to 90 m wide with an average depth of about 5 m
- Aligned with slightly more than 170 buildings
- A sort of street canyon whose bottom surface is not solid but water
- Public transport provided by water boats and private water taxis expected to be the major source of pollution

► are traditional air quality models suitable for this special urban environment? ◄

► could classical CFD modelling be applied to Venice where most street canyons are represented by the city channels? ◄

MODELLING AND WIND TUNNEL SETUP

Simplified geometry of the Canal Grande

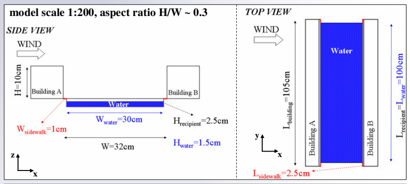
General CFD setup

- Reynolds Stress Model
- Inlet:

$$\frac{U(z)}{U_{ref}}=\left(\frac{z}{z_{ref}}\right)^{0.35} \quad k=\frac{u_*^2}{\sqrt{C_\mu}}\left(1-\frac{z}{\delta}\right) \quad \varepsilon=\frac{u_*^3}{\kappa z}\left(1-\frac{z}{\delta}\right)$$

$u_* = 0.44 ms^{-1}$, $\delta = 1m$, $U_{ref} = 8.13 ms^{-1}$, $z_{ref} = 0.75m$ (model scale)

- symmetry conditions at top and lateral sides; pressure-outlet condition, second order upwind discretization scheme
- Ground level source emitting N_2O, emission rate $Q = 10 g/s$



No water-street canyon

- **3D steady-state simulations** (FLUENT 6.3.26)
- Mesh: one million hexahedral elements (smallest cell dimensions δxmin = 0.05H, δymin = 0.1H, δzmin = 0.025H)
- Mean gas concentrations $c^+ = \frac{c\,U_{ref}\,H}{Q/l}$ where l = 50cm is the line source length

Water-street canyon

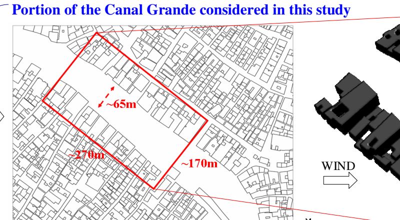
- **2D unsteady simulations, multiphase VOF model (Volume of Fluid)**
- Conservative numerical settings for typical VOF model:
- •Explicit (VOF scheme), SIMPLE for pressure-velocity coupling, modified HRIC or Geo-reconstruct for volume fraction
- Mesh: 500,000 hexahedral elements

δx = 0.0125H (horizontal), δy = 5e-05H (vertical) at air-water interface

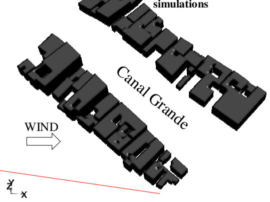
- **Wind tunnel experiments** (Department of Technology and Built Environment of the University of Gävle, Sweden)
- Small plastic floating objects employed to visualize the water movement at several approaching wind speeds and aspect ratios (~ 0.3, ~ 0.6 and ~ 0.9)

Real scenario – Canal Grande

Portion of the Canal Grande considered in this study



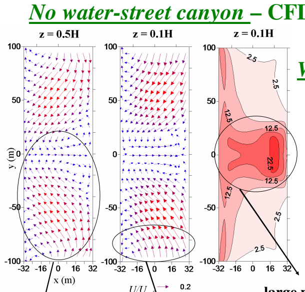
Sketch of the Canal Grande portion geometry considered in CFD simulations



- Only the ***no water case*** simulated by applying the same set-up used in the idealized street canyon
- Main geometry details taken from a digital CAD file available from Insula S.p.a. (Società per la Manutenzione Urbana di Venezia)
- Tallest building (Hmax ~ 26m). Mesh: one million elements (full scale cell dimensions δxmin = δymin = 2m, δzmin = 0.5m till a height of 5m)

RESULTS

No water-street canyon – CFD



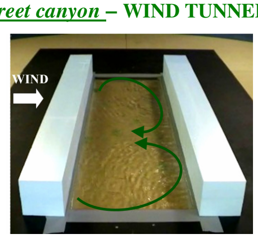
two symmetric corner vortices close to the ground

large pollutant concentrations

in the middle, 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.

Air moves along the wind direction in the middle of the canyon leading to large concentrations at the windward

Water-street canyon – WIND TUNNEL



floating objects move through two symmetric vortices at all the wind speeds and aspect ratios considered.

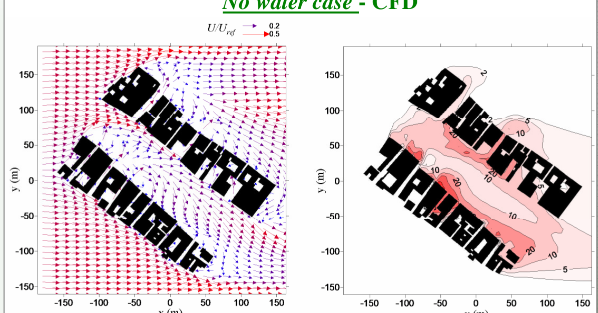
Contrary to what happens in the *no water-street canyon* case, in the *water* case floating objects reaching the middle of the canyon move opposite to the wind direction

Water-street canyon – CFD

Preliminary indications (work in progress):

- Unsteady
- Mesh at air-water interface: δx = 0.0125H (horizontal), δy = 5e-05H (vertical)
- Time step: not larger than ~1e-05
- Geo-reconstruct for volume fraction

No water case - CFD



- flow channelled along the Canal Grande
- an 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

CONCLUSIONS

- CFD simulations suggested 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**
- 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**
- *Further testing are 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. Alexandro Palmieri from University of Bologna for his help in CFD simulations.