THE INFLUENCE OF BUOYANCY ON FLOW AND POLLUTANT DISPERSION IN STREET CANYONS

Riccardo Buccolieri[1], Beatrice Pulvirenti[2], Silvana Di Sabatino[1], Rex Britter[3]

[1]Dipartimento di Scienza dei Materiali, University of Salento, Lecce, Italy
[2]Dipartimento di Ingegneria Energetica, Nucleare e del Controllo Ambientale, University of Bologna, Bologna, Italy
[3]Department of Engineering, University of Cambridge, Cambridge, UK

Abstract: In this paper, the effect of buoyancy on flow and pollutant dispersion within street canyons is studied by means of computational fluid dynamics simulations. We consider a neutral boundary layer approaching a 3D street canyon assuming a wind direction perpendicular to the street canyon. The Boussinesq hypothesis for incompressible fluids is chosen for modelling buoyancy. We distinguish three cases: leeward, ground and windward wall heating. Thermal effects on both the flow and dispersion are investigated for several Richardson numbers. The analysis focuses on the influence of street canyon geometry on flow and temperature distribution, by considering different aspect ratios W/H canyon between 0.5 and 2, where W is the width and H the height of the street canyon. Three-dimensional effects are observed, depending on L/H, where L is the length of the canyon. Three dimensional effects become negligible for aspect ratio L/H larger than 20. Results obtained for the case with a large Richardson number show that dispersion patterns in a street canyon differ substantially the isothermal case. In case with windward heating large concentration values are found close to the windward wall. Our findings can be of interest for many urban environment applications in which natural ventilation and thermal comfort are being of concern.

Key words: *Buoyancy effects, pollutant concentration, CFD simulations, street canyon, Richardson number.*

1. INTRODUCTION

Due to rapid urbanization and industrialization, streets have become one of the most common areas where both pedestrian and traffic density are usually high. Until now, interest in the study of flow and pollutant dispersion in urban street canyons has rapidly increased. To achieve a better understanding of street canyon flow and pollutant dispersion and to provide useful guidelines for urban air quality and planning, a large number of studies have been carried out through field measurements, wind tunnel experiments and numerical model simulations (Ahmad et al., 2005; Vardoulakis and Bernard, 2003). They have identified many factors affecting flow and dispersion in street canyons, such as the ambient wind speed and direction, the building geometry and so on.

Thermal effects are another important factor affecting street canyon flow and pollutant transport. In urban areas, those effects are generally due to solar radiation on building walls and ground surface that heats up air in the vicinity. The role of buoyancy forces within street canyon is particularly relevant under calm wind conditions, when downward inertial forces are often compensated by an upward flow due to the buoyancy.

Numerical studies of buoyant flows are less numerous in the literature with respect to those ones under isothermal conditions. Most numerical investigations are limited to two-dimensional simulations using steady Reynolds Averaged Navier–Stokes equations (RANS) models (e.g. Xie et al., 2007) or unsteady RANS models (Sini et al., 1996; Kim and Baik, 2001). Two-dimensional studies usually do not account for the highly three-dimensional flow fields present in real urban street canyons of finite length. Recently, using a three-dimensional, steady RANS model, Tsai et al. (2005) have examined thermal effects of heated building walls on flow and pollutant dispersion in a street canyon of aspect ratios H/W=0.8 and L/W=3 (where H is the height, W the width and L the length of the street canyon). They showed that the vortex line that connects the centres of the cross sectional vortices meanders in the street canyon. Only few studies report on three-dimensional numerical simulations of thermal effects. Coupling thermal and dispersion aspects in street canyons are rare in the literature due to the high computational cost (Tsai et al., 2005; Baik et al., 2007; Kang et al., 2008).

In this paper, we investigate the impact of ground and wall heating on flow and pollutant dispersion in a street canyon by means of the computational fluid dynamics (CFD) code FLUENT by employing the standard k- model for flow and the advection-diffusion scheme for dispersion. Three street canyon aspect ratios have been considered, which are W/H=0.5, 1 and 2. To avoid unrealistic three-dimensional effects, depending on L/H, a ratio L/H=20 has been chosen after performing several tests. At the inlet section, ambient air temperature at ground level has been assumed to be equal to 27°C. Cases with and without heating at different street canyon boundaries such as heating at ground level, leeward and windward sides are analysed by imposing the corresponding wall temperature to 37°C. Three different inlet velocities corresponding to three different Richardson numbers have been considered.

2. MODELLING SETUP

FLUENT simulations have been carried out by considering an approaching neutral boundary layer flow. The computational domain is a parallelepiped with dimensions 320 m in the x direction, parallel to the flow direction, 310 m in the y direction and 80 m in the vertical z direction. The buildings are two parallelepipeds with dimensions 10 m by 200 m (L) by 10 m (H). The width W of the canyon varies from 5 to 20 m, in order to give aspect ratios in the range 0.5<W/H<2. The choice of the building dimensions has been the result of a wide number of preliminary tests devoted to understand the dimensionality of the problem. We have found that for L/H<15 the velocity field is strongly 3D, as discussed in the next section. In this paper, L/H=20 for all the aspect ratio is considered.

The dimensions of the computational domain gives an appropriate mesh size for the required flow detail and run time. The computational domain was built using structured elements with a finer resolution close to the ground and to the walls within the canyon. A finer refinement with respect to the isothermal simulations was required to capture thermal gradients near the walls. Several tests have been performed to verify grid size independence with increasing mesh numbers. The final number of the computational cells used is about 1 million for all cases. The smallest dimension of the elements, in the region near the heated walls is 0.25 m in the direction perpendicular to the wall and 0.5 m in the other directions.

Reynolds averaged equations with a standard k- turbulence closure (Launder and Spalding, 1974), together with the Fourier equation have been considered for flow and temperature, respectively. The Boussinesq approximation has been assumed. Buoyancy effects are studied by means of the dimensionless Richardson number

$$Ri = \frac{\left[g\left(T_w - T_a\right)H\right]}{T_a u^2} = \frac{Gr}{Re^2} \qquad (1)$$

where g is the gravitational acceleration, T_W is the wall temperature (leeward, ground and windward of the street canyon), T_a is the ambient air temperature, H is the building height and u is the reference velocity (wind velocity of the undisturbed flow). At the inlet section, ambient air temperature at ground level has been assumed to be T_a=27°C. Cases with and without extra heating at ground level, the leeward and windward sides of the street canyon are analysed (as shown in Fig. 1) by imposing the corresponding wall temperature to T_W=37°C.

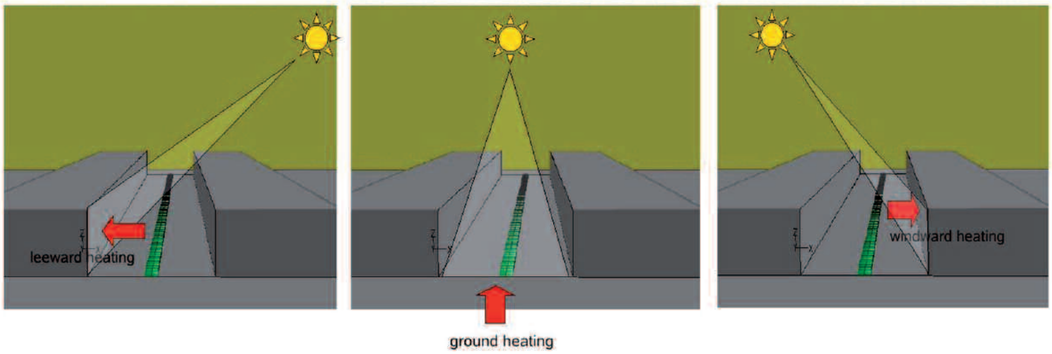

Figure 1. Simulation of wall heating in an urban street canyon. Leeward heating (left), ground heating (middle) and windward heating (right).

The inlet boundary condition is a neutral boundary layer profile similar to that used in our previous works (Di Sabatino et al. 2007, 2008), where the friction velocity u* has been modified to ensure specific wind velocities of the undisturbed flow. Three different u, namely 1 ms^{-1}, 2 ms^{-1} and 3 ms^{-1}, have been considered to give Ri=3.27, 0.82 and 0.36, respectively. Inlet turbulent kinetic energy and dissipation rate profiles are specified as Di Sabatino et al. (2007, 2008). Symmetry boundary conditions are specified on the top and lateral sides of the computational domain.

Dispersion modelling within FLUENT is done by using a diffusion term in the pollutant transport equation as follows

$$J = -\left(\rho D_t + \frac{\mu_t}{Sc_t}\right)\nabla Y \qquad (2)$$

where D is the diffusion coefficient for pollutants in the mixture, $\mu_t = ½(C_\mu k^2/\varepsilon)$ is turbulent viscosity, Y is the mass fraction of pollutants, ρ is the mixture density. $Sc_t=\mu_t/(½D_t)$ is the turbulent Schmidt number, where D_t is the turbulent diffusivity. A line source has been simulated by separating a volume in the geometry on the ground at the centre of the street canyon and by setting a source term for this volume. The emission rate of a passive pollutant is set at Q=10gs^{-1} over the length of the release.

3. RESULTS

A large number of preliminary tests has shown that for L/H<15 the velocity field is strongly 3D. Figure 2 shows the contours of the transversal component of the velocity in a horizontal plane at z/H=0.5, for a case with L/H=10 and W/H=1. Figure 2 shows that, in the cases of leeward and ground heating, a large amount of air enters in the canyon from the lateral openings. Then, the problem becomes strongly three-dimensional, and the expected single vortex structure is not observed even in the middle of the canyon. To avoid these 3D effects we have used L/H=20 for all W/H aspect ratios considered.

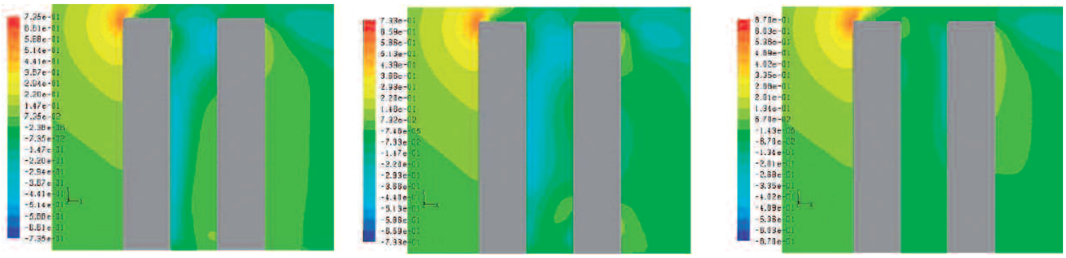

Figure 2. Y component of velocity field. Leeward heating (left), ground heating (middle) and windward heating (right).

For sake of brevity in the following section will discuss those cases for which mechanical effects are weaker being wind velocity small. In our case it corresponds to Ri=3.27.

Velocity results

Figure 3 shows path lines coloured by the vertical component (z-component) of wind velocity on a vertical plane in the middle of the canyon. In the leeward heating case a weak dependence on the aspect ratio is observed as the vortex within the canyon is enhanced by the buoyancy. In the ground heating case we can observe that the vortex becomes weaker as the aspect ratio increases; a larger and larger vertical wind velocity component in the central region could break the vortex at smaller aspect ratios. In the windward heating case the classical counter-rotating vortex is observed for W/H=2. Many authors have found a similar velocity distribution within the canyon in the case of windward heating. The vortex becomes dominant as the aspect ratio W/H decreases and for W/H=0.5. This suggests that the 'cold clockwise' vortex is suppressed by the counter-rotating vortex which in this case occupies the whole region of the canyon.

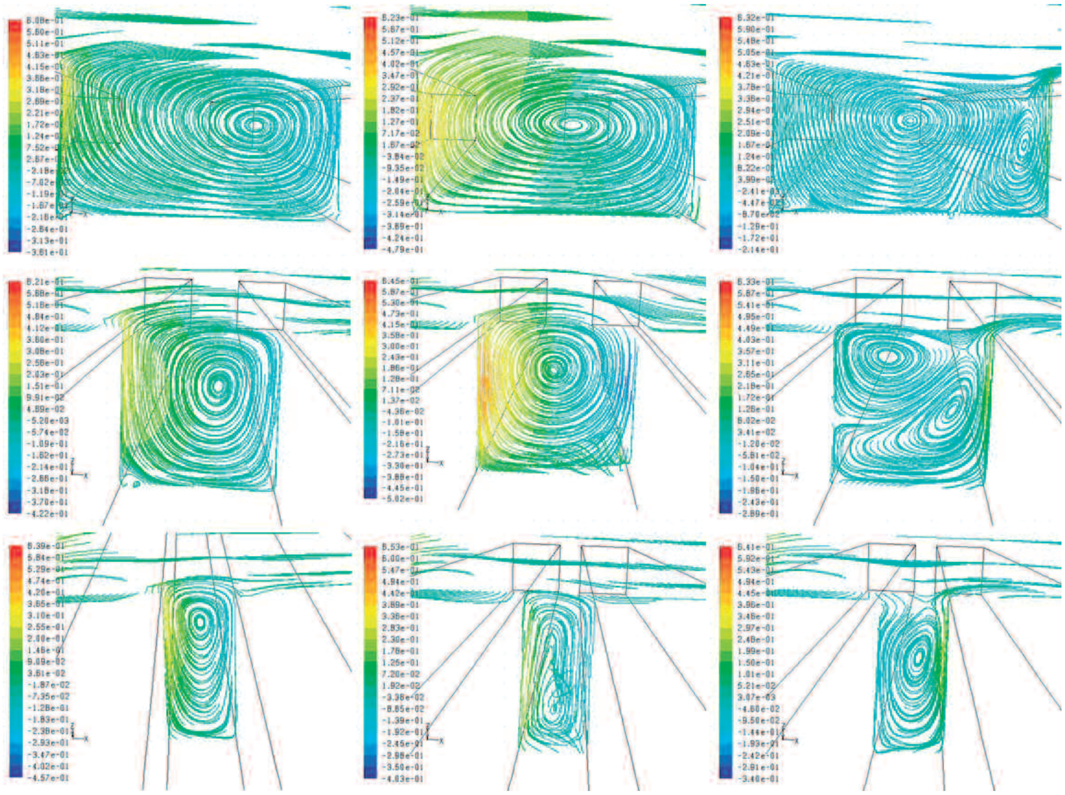

Figure 3. Path lines for Ri=3.27. Aspect ratios: W/H=2 (top), W/H=1 (middle) and W/H=0.5 (bottom). Leeward heating (left), ground heating (middle) and windward heating (right).

Temperature results

Figure 4 shows temperature contours on a vertical plane in the middle of the canyon, for the case Ri=3.27. These results explain changes observed in the velocity profiles discussed in the previous section.

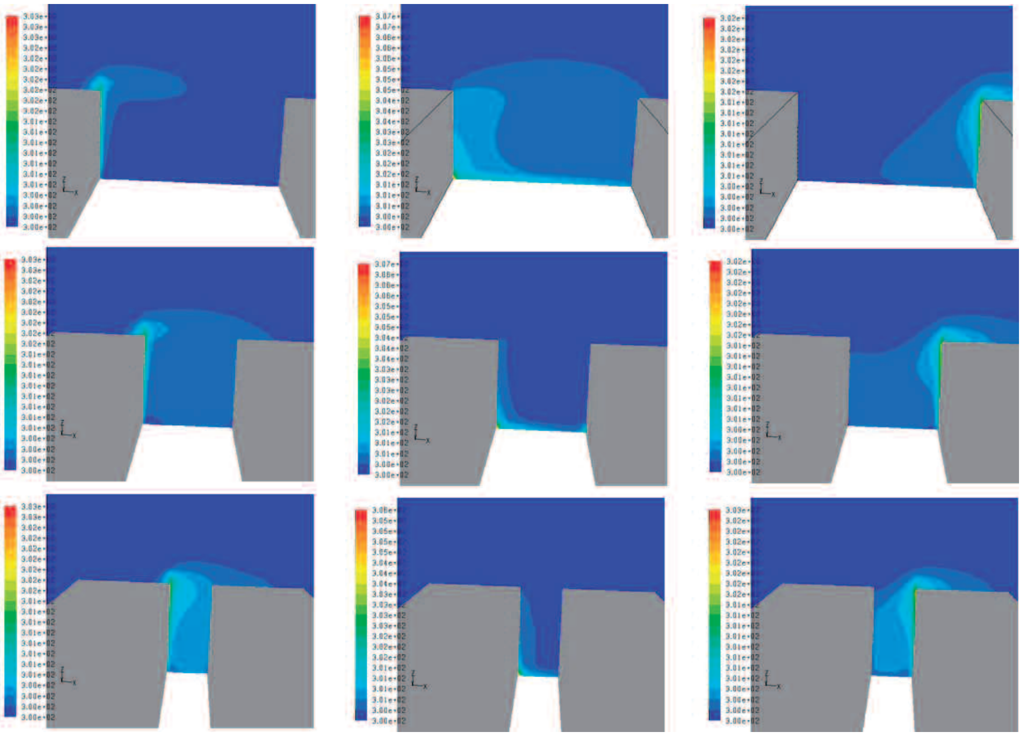

Figure 4. Temperatures contours for Ri=3.27. Aspect ratios: W/H=2 (top), W/H=1 (middle) and W/H=0.5 (bottom). Leeward heating (left), ground heating (middle) and windward heating (right).

Dispersion results

For the study of dispersion we observe a stronger dependence of the results on numerical discretization. Figure 5 shows the grid used for simulations discussed in the previous sections. This grid is sufficiently fine to give dynamical and thermal boundary layers resolved over the top of the buildings and within the canyon.

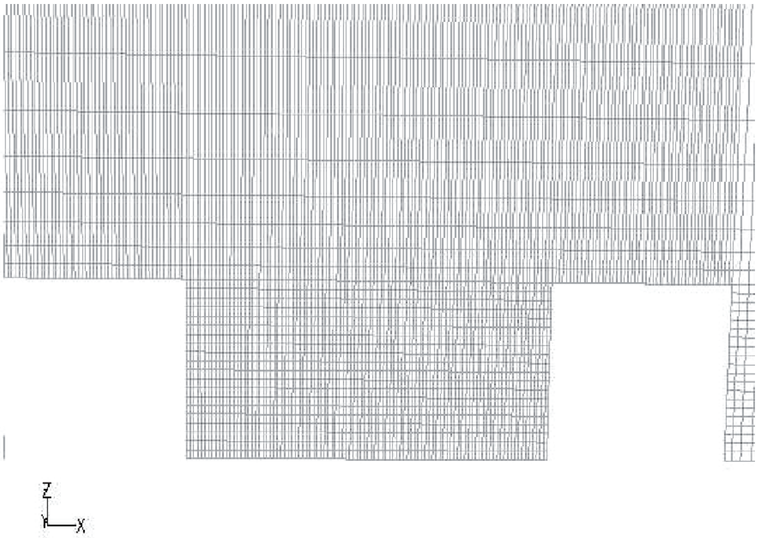

Figure 5. Grid resolution within the canyon and on top of buildings.

After introducing the dispersion model in our simulations we observed that this grid is not sufficient to correctly capture thermal and dispersion effects together and further refinement is still required. Figure 6 shows how the velocity fields are modified by the introduction of dispersion modelling. The figure refers to the case W/H=2, windward heating, Ri=3.27. Without dispersion (left in the figure) the two vortices described in the previous sections are obtained within the canyon. With dispersion (right in the figure) the flow structure within the canyon has been unrealistically altered: both vortices have been destroyed and a large vertical velocity component is observed over the top of the buildings. The strong dependence of results on the grid used maybe due to the particular CFD model used and may not reflect a particular feature be of a CFD calculation. Further work is undergoing to investigate this aspect.

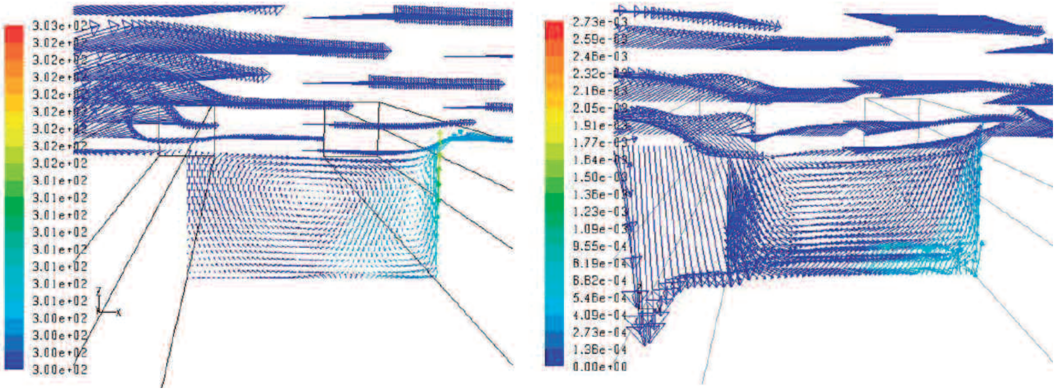

Figure 6. Comparison of velocity field without dispersion (left) and with dispersion (right). Vectors are coloured by temperature on the left and by CO concentration on the right.

4. CONCLUSIONS

The effects on flow and pollutant dispersion in street canyons of different aspect ratios with heated walls have been studied. The results show that with leeward heating, the canyon vortex is enhanced and its centre is shifted towards the windward causing lower concentrations to occur. In the case of windward heating, convective flow leads to the introduction of cooler air from the bottom of the canyon forming a second vortex close to the windward wall. In this case larger pollutant concentrations near the windward wall and within the whole street canyon are found.

REFERENCES

Ahmad, K., M. Khare and K.K. Chaudhry, 2005: Wind tunnel simulation studies on dispersion at urban street canyons and intersections—a review. *J. Wind Eng. Ind. Aerod.*, **93**, 697–717.

Baik, J.-J., Y.S. Kang and J.J Kim, 2007: Modeling reactive pollutant dispersion in an urban street canyon. *Atmospheric Environment*, **41**, 934–949.

Di Sabatino, S., R. Buccolieri, B. Pulvirenti and R. Bitter, 2008: Flow and pollutant dispersion in street canyons using FLUENT and ADMS-Urban. *Environmental Modeling and Assessment*, **13**, 369–381.

Di Sabatino, S., R. Buccolieri, B. Pulvirenti and R. Bitter, 2007: Simulations of pollutant dispersion within idealised urban-type geometries with CFD and integral models. *Atmospheric Environment*, 41, 8316–8329.

Kang, Y.S., J.J. Baik and J.J. Kim, 2008: Further studies of flow and reactive pollutant dispersion in a street canyon with bottom heating. *Atmospheric Environment*, **42**, 4964–4975.

Kim, J.J. and J.J. Baik, 2001: Urban street-canyon flows with bottom heating. *Atmospheric Environment*, **35**, 3395–3404.

Launder, B. E. and D.B. Spalding, 1974: The Numerical Computation of Turbulent Flows. *Computer Methods in Applied Mechanics and Engineering*, **3**, 269-289.

Sini, J.F., 1996: Pollution dispersion and thermal effects in urban street canyon. Atmospheric Environment, 30, 2659–2677.

Tsai, M.Y., K.S. Chen and C.H. Wu, 2005: Three-dimensional modeling of air flow and pollutant dispersion in an urban street canyon with thermal effects. *Journal of the Air and Waste Management Association*, **55**, 1178–1189.

Vardoulakis, S. and E. Bernard, 2003: Modeling air quality in street canyons: a review. *Atmospheric Environment*, **37**, 155–182.

Xie, X., C.H. Liu and D.Y.C. Leung, 2007: Impact of building facades and ground heating on wind flow and pollutant transport in street canyons. *Atmospheric Environment*, **41**, 9030–9049.