

TEST CASE OPTIMIZATION OF AN URBAN SECTION

Alessandra Borsani^{*}, Silvia Maltagliati^{*}, Francesco Martelli^{*}, Lorenzo Procino^{},**

^{*}Dipartimento di Energetica S.Stecco

Università degli Studi di Firenze, via s. Marta 3, 50139 Firenze, Italy

e-mails: alessandra.borsani@pin.unifi.it

silvia.maltagliati@unifi.it

francesco.martelli@unifi.it

^{**}CRIACIV Boundary Layer Wind Tunnel

c/o PIN, p.za dell'Università 1, 59100 Prato, Italy

e-mail: lorenzo.procino@pin.unifi.it

Keywords: CFD, urban environment, PIV, pressure, modeling

1 INTRODUCTION

Urban planning in nowadays targeted to reach air quality objectives also, and related health risks monitoring. The diagnostic and forecasting tools to assess ambient pollution must be reliable and able to evaluate different scenarios of land use and anthropogenic activity.

The most common way of dealing this issue has been through operational non-CFD models, also addressed as “integral” (Di Sabatino et al. 2007), which are not able to predict flow, dispersion and the resulting concentrations in complex urban or industrial areas. In this case the use of the CFD techniques to model flow and pollutant dispersion could be more appropriate since they are able to simulate the local effects of the buildings that influence significantly the flow (Riddle 2004, Holmes 2006).

However, the trustworthiness of this approach is still dependent on the expertise of the code user, due to the difficulty of setting up a good simulation (in terms of geometry, mesh and parameter definition) and to the lack of a standard procedure to follow in these applications.

The main purpose of this study is to find an optimal test case for a CFD simulation, performed with a well known code, in order to have a standard numerical benchmark to be used for testing simulations in cases similar with the test case. As suggested by the common experience of mechanical engineering, the sensitivity of a CFD tool to the environment parameters can be estimated through an experimental campaign in a wind tunnel of a physical model representing the scaled domain.

In this paper the domain is a group of buildings in different geometrical configurations, typical of an Italian urban environment, immersed in a flow field under neutral conditions. The test case has been set in the Boundary Layer Wind Tunnel of CRIACIV, (Florence, Italy), where the physical model of the buildings has been placed in the test section and flow visualization and flow velocimetry (PIV techniques) has been performed. Pressure tabs have

been equipped as well in order to measure the pressure coefficient in the symmetry plane of the physical model.

The CFD simulations have been performed with the commercial code FLUENT in the full-scale domain.

In the following paragraphs both the experimental and the computational set up are presented.

2 DEFINITION OF THE DOMAIN

2.1 Geometry of the group of buildings

The domain is a typical urban environment, where a small building is set upwind to a higher one. The main concern is to evaluate the flow patterns of streamlines coming from the small building and affected by the higher one. In fact this work is the preliminary study for the modeling of the dispersion pattern of emissions from a source placed on the small building and diversely affected by the higher building.

In particular, the size of the ‘blocks’ representing the buildings in the CFD simulation are realistic for a urban environment from Florence, in which buildings range from 1 floor (3m high) up to 5 floors (15 m high). Different models have been set up by varying the front building height: 3 m, 9 m and 15 m (Figure 1, Figure 2, Figure 3 and Figure 4).

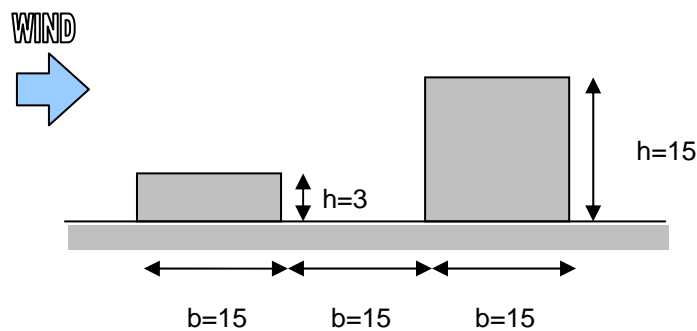


Figure 1 : Sketch of the model 1, plane x-z.

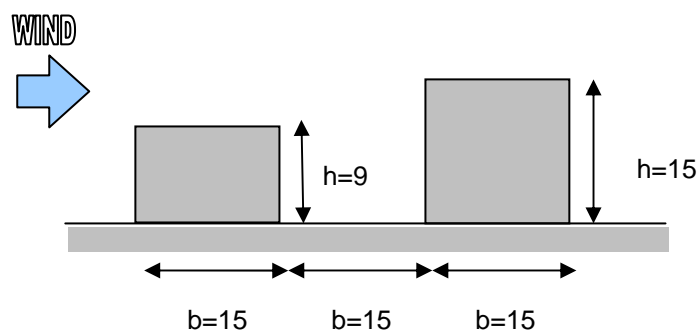


Figure 2 : Sketch of the model 2, plane x-z.

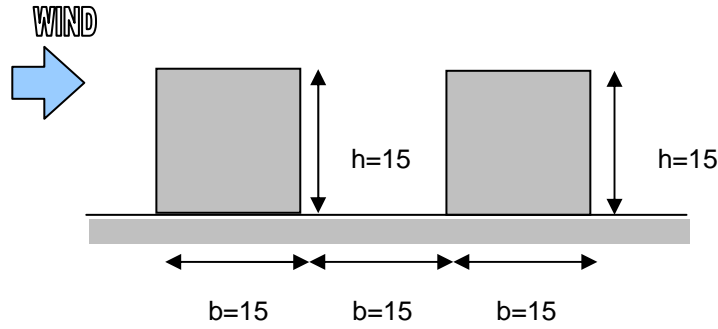


Figure 3 : Sketch of the model 3, plane x-z.

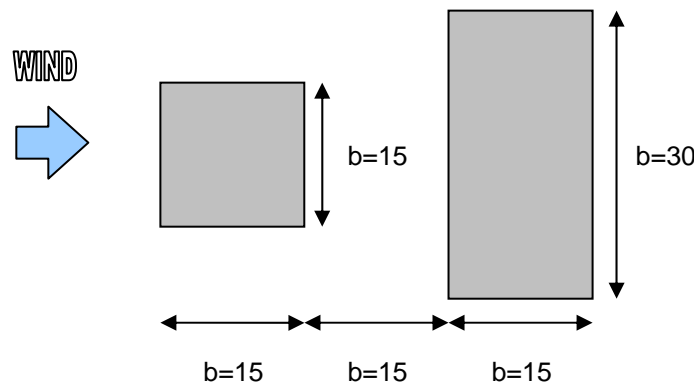


Figure 4 : Sketch of the model 1, 2 and 3, plane x-y.

The distance between the two adjacent buildings is set always 15 m.

2.2 Meteorological neutral conditions

As part of the domain, the meteorological conditions typical of the urban environment must be set, with appropriate dynamical scaling in the wind tunnel and CFD simulation. The main constraint is about the wind tunnel boundary layer turbulence profile, which can be produced through mechanical devices only. This leads to consider the neutral condition of the atmospheric boundary, where almost the only turbulent kinetic energy generation mechanism is mechanical, associated with wind shear and surface stress (Stull, 1997). Applying the neutral boundary layer similarity theory, the wind, turbulent kinetic energy (TKE) and turbulence dissipation rate profiles $U(z)$, $k(z)$ and $\varepsilon(z)$ are foreseen as follows:

$$U(z) = \frac{u^*}{k} \ln \frac{z}{z_0} \quad (1)$$

$$k(z) = const = \frac{u^{*2}}{\sqrt{C_\mu}} \quad (2)$$

$$\varepsilon(z) = \frac{u^{*3}}{k \cdot z} \quad (3)$$

where u^* is the friction velocity, z_0 is the aerodynamic roughness length, k is the von Karman's constant and C_μ is an empirical constant.

Moreover, it must be considered that the wind velocity is rarely above 4 m/s in Florence.

3 WIND TUNNEL TESTS

3.1 Physical model

The models 1, 2 and 3 have been reproduced in small scale 1:150 (Fig. 5). The models have been placed into an incoming wind characterized by a logarithmic profile of velocity generated in a section of development of the boundary layer by spires at the inlet and 5 cm height roughness elements at the floor (Fig. 6).

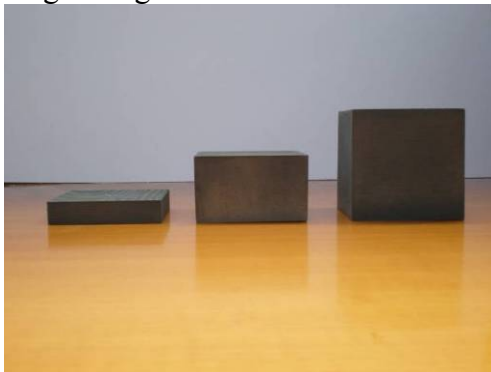


Figure 5 : Models of the front building with different heights: 2 cm, 6 cm and 10 cm.



Figure 6 : Model into the wind tunnel model

Pressure tabs have been positioned in the symmetry plane of the model at a mutual distance of 2 cm on the floor and of 1 cm on the buildings.

3.2 Measuring set-up

The characterization of the incoming wind profile has been revealed through velocity measurement with pitot-prandtl tube and a hot wire anemometer at many heights in the middle plane of the test section.

Measurements of the flow field in the middle plane have been made through Particle Image Velocimetry (PIV) technique.

The pressure measurements were made through tabs, positioned on the floor and on the sides of the buildings, connected to miniaturized pressure transducers of 'PSI Pressure System'.

3.3 Incoming wind profile

Following is reported the incoming wind profile measured 0.5 m before the blocks, both in terms of mean wind speed (Fig.7) that in terms of TKE (Fig.8, blue quadrates). The TKE has been measured also for three heights in a second plane, positioned 0.7 m after the first, in order to verify if occurs a decay of energy through the test section (Fig.8, purple triangles).

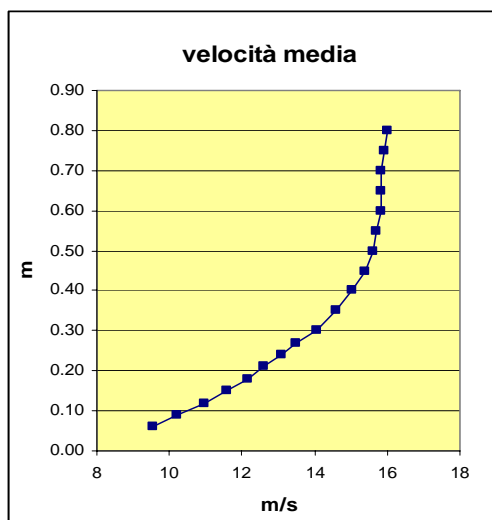


Figure 7 : Measured profile of mean velocity

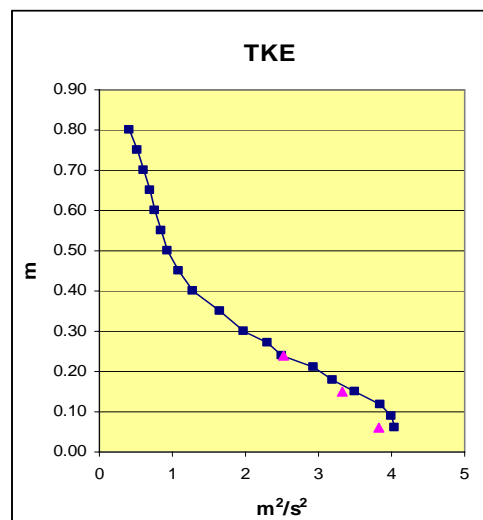


Figure 8 : Measured profile of TKE

4 CFD SIMULATION

4.1 Simulation domain

The domain of the CFD simulation has been chosen with respect to the COST ACTION 732. The lateral extension of the domain is 75 m for each side, equal to 5H, where H is the height of the two buildings; the total largeness of the domain is 180 m.

The inflow boundary as well is positioned at a distance of 5H=75 m from the built area and the outflow boundary is 180 m behind the built area equal to 12H. The recommended distance of the outflow boundary is 15H but can be reduced with for steady RANS simulations with open boundary conditions, paying attention to avoid reverse flow entering the domain trough the outlet boundary. In this case the boundary condition is constant pressure and there is no reverse flow. The total length of the domain is 300 m.

The recommended height of the domain is 5H=75 m but it is been raised to 100 m in order to avoid acceleration of the flow. The blockage is 2.5%.

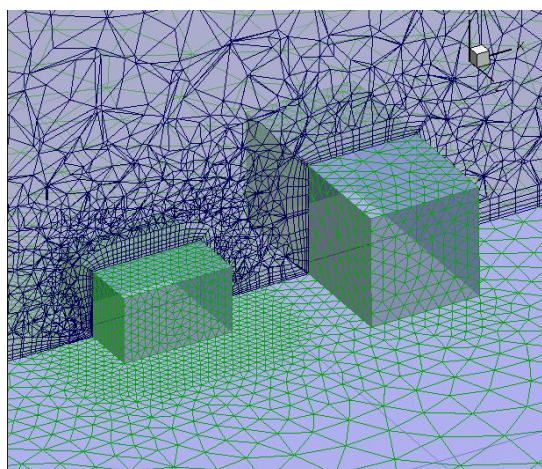


Figure 9 : Mesh of the model 3

The mesh is a hybrid one composed by tetrahedral elements and by five layers of prismatic cells near the solid walls.

The quality of the grid has been tested through a mesh refinement procedure.

4.2 Boundary conditions

The inflow condition is a velocity profile equal to the one measured in wind tunnel (§3.3). The profile has been approximated with the following logarithmic law:

$$U(z) = \frac{u^*}{k} \ln \frac{z}{z_0} \quad (4)$$

In which z_0 is the roughness length, u^* is the friction velocity and k is the von Karman's constant.

The wind tunnel profile (in scale 1:150) is characterized by: $z_0=0.00214$ m, while in the CFD profile z_0 has been multiplied by the scale factor and is therefore equal to 0.32 m; the Re similarity has not been maintained ($Re_{\text{model}} \neq Re_{\text{real}}$), assuming the flow structures independent from Re number, due to the sharp corners of the blocks [Simiu, Scanlan, 1996]. The results have been all adimensionalized by the mean velocity at the height of the top of the second building.

The profile of TKE has been set after the literature [Wilcox]:

$$k(z) = \frac{u^{*2}}{\sqrt{C_\mu}} \quad (5)$$

where $C_\mu = 0.09$.

As evident from figure 8, the measured profile differs from the theoretical form, that will be tested in a following work.

The turbulent dissipation rate profile in the *standard* κ - ε model has been set as eq. 3 [Simiu Scanlan, Wilcox], consequently the specific dissipation rate in the *standard* κ - ω model is:

$$\omega(z) = \frac{u^*}{k \cdot z \cdot \sqrt{C_\mu}} \quad (6)$$

The outflow boundary condition is constant pressure. The lateral and superior side have a symmetry condition.

5 RESULTS AND DISCUSSION

The CFD simulations performed with the boundary conditions derived from the wind tunnel set up have been validated through the comparison of the computed variables with the measurements. In fact, the flow field in the test section has been visualised through the PIV and the pressure profiles have been measured on the solid walls of the buildings and the surrounding ground.

5.1 Flow structures

In the following figures (From fig. 10 to 18) the isovelocity maps in the zone between the two buildings are shown. For each configuration CFD results (k - \square and k - ω turbulence models) and PIV measurements are reported. In the same pictures streamlines are visible as well.

Flow visualization for the first model:

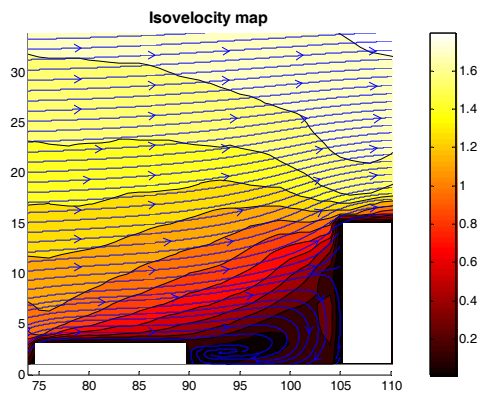


Figure 10 : model 1, *standard $\kappa-\epsilon$*

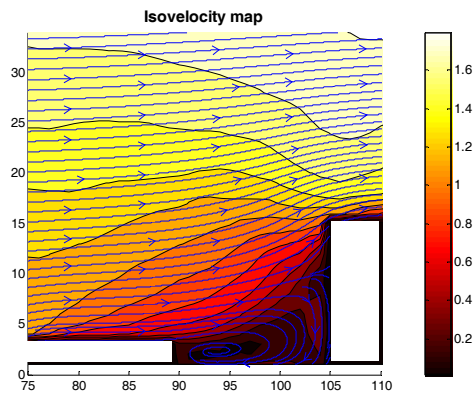


Figure 11 : model 1, *standard $\kappa-\omega$*

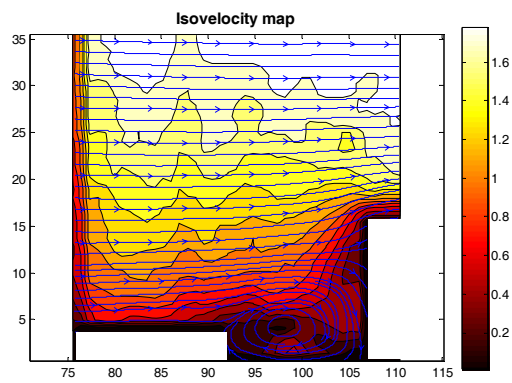


Figure 12 : model 1, PIV measurement

Flow visualization for the second model:

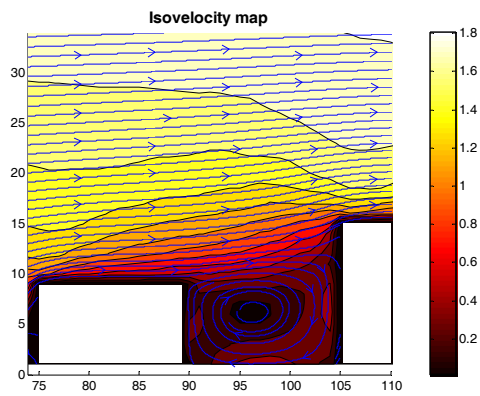


Figure 13 : model 2, *standard $\kappa-\epsilon$*

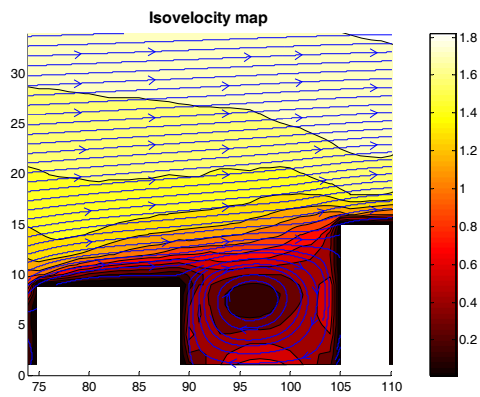


Figure 14 : model 2, *standard $\kappa-\omega$*

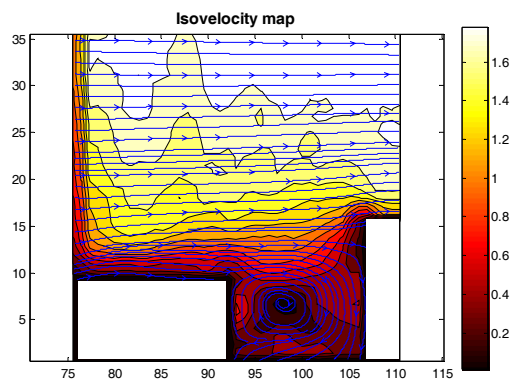


Figure 15 : model 2, *PIV measurement*

Flow visualization for the third model:

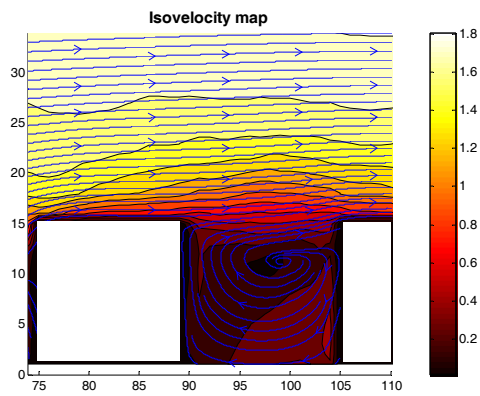


Figure 16 : model 3, *standard $\kappa-\epsilon$*

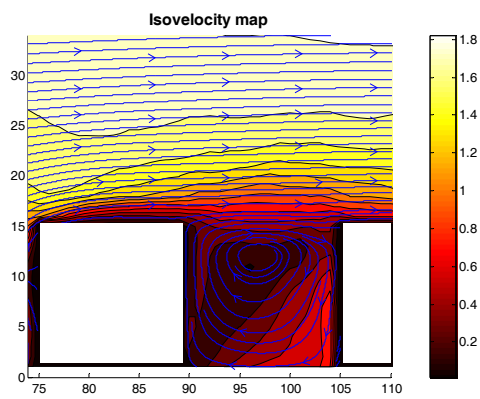


Figure 17 : model 3, *standard $\kappa-\omega$*

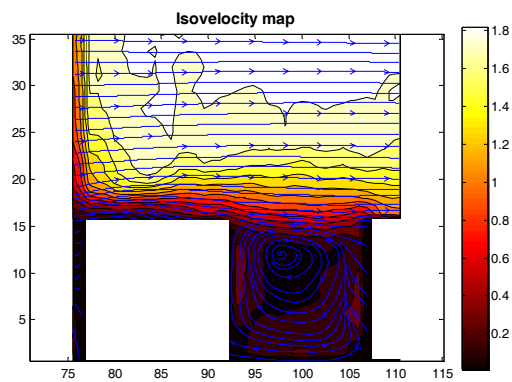


Figure 18 : model 3, PIV measurement

5.2 Pressure

The pressure coefficients of the CFD results and of the wind tunnel pressure measurements are compared in the following pictures (Fig. 17, 18 and 19). The pressure values have been nondimensionalized through the value of the pressure measured in the middle of the front side (upwind) of the first building. In the second and third pictures are presented the results of the numerical model with standard k-e and standard k-w turbulence model respectively; in the fourth are visible the pressure coefficients measured during wind tunnel tests; while in the first picture all this configurations are presented together.

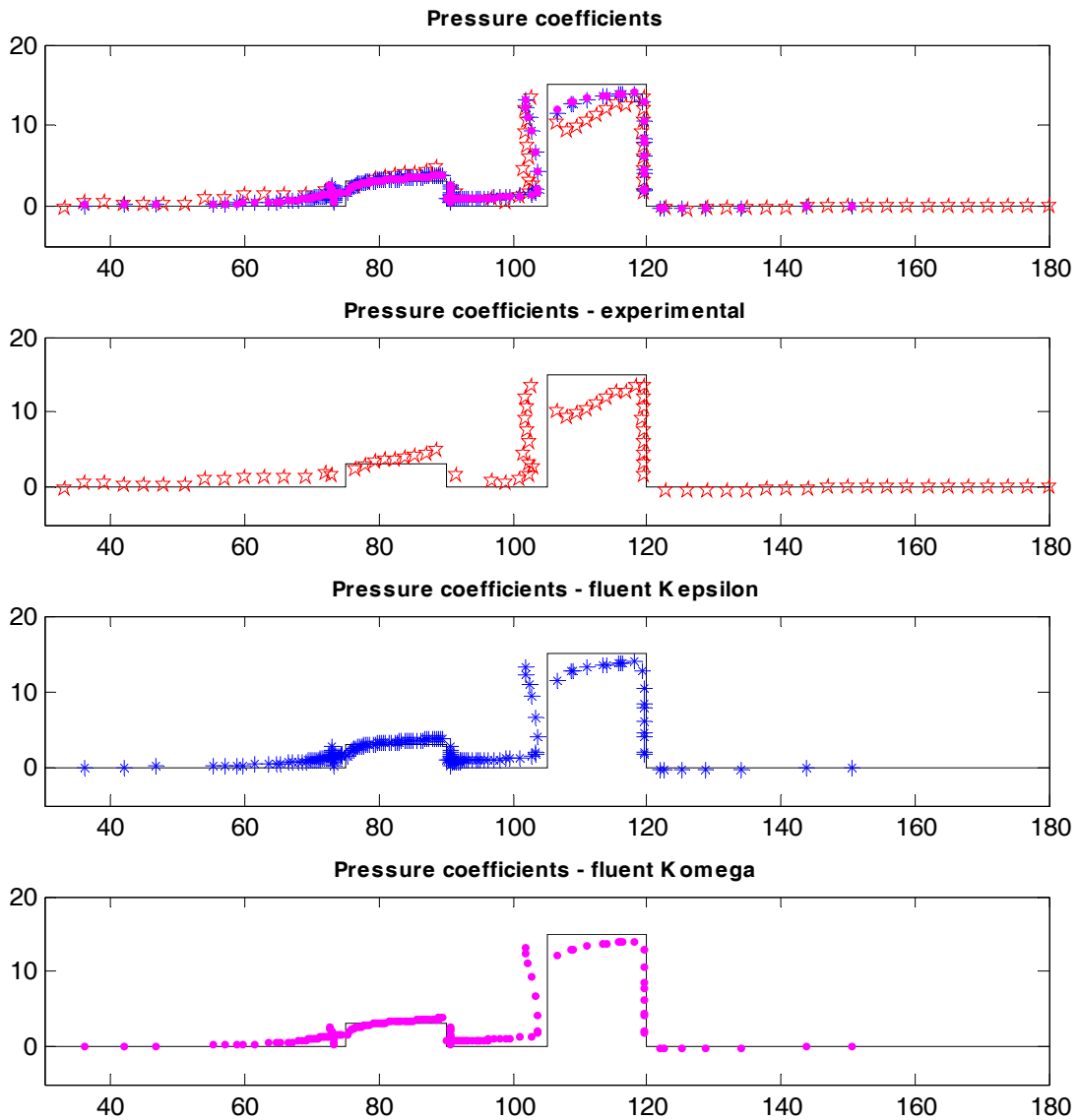


Figure 19 : model 1, pressure coefficients

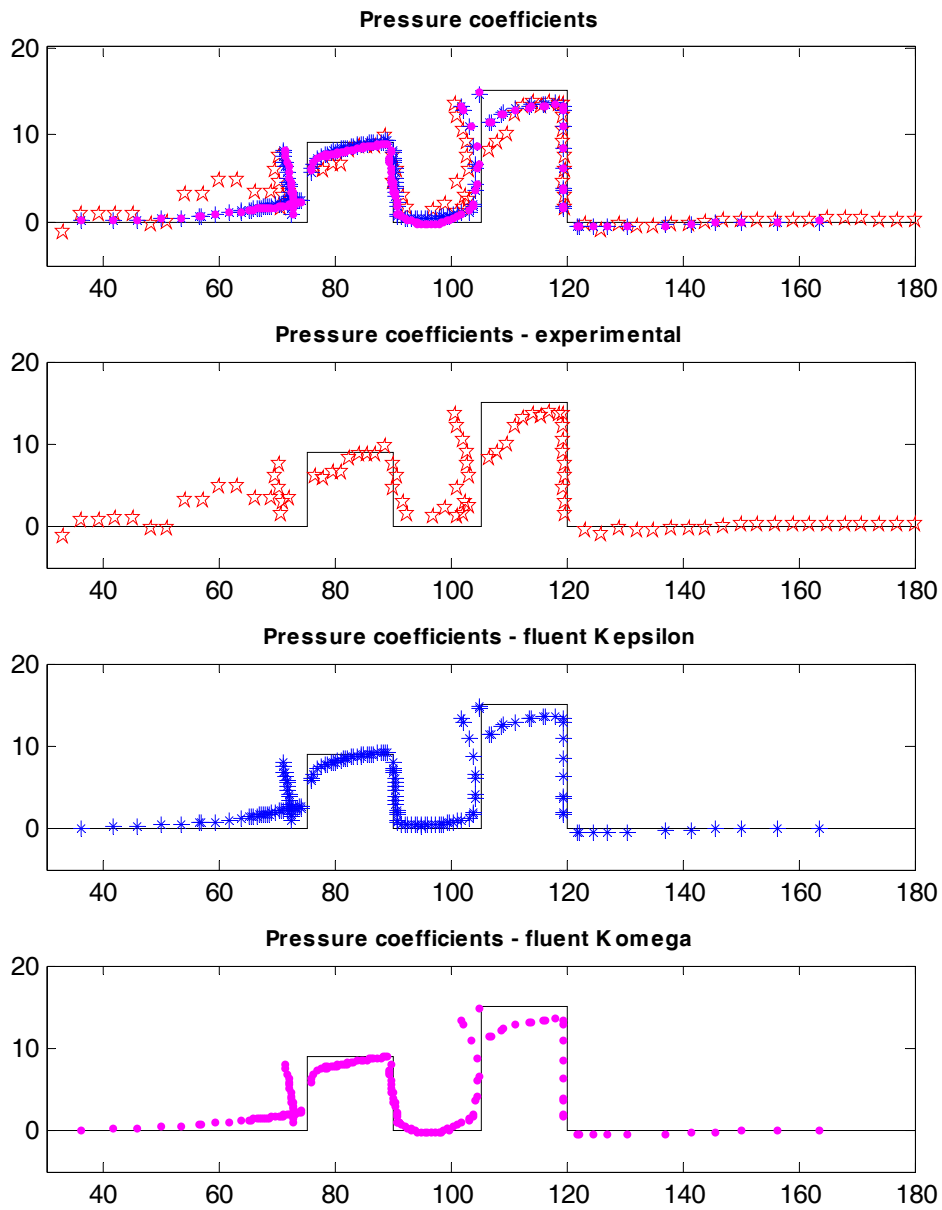


Figure 20 : model 2, pressure coefficients

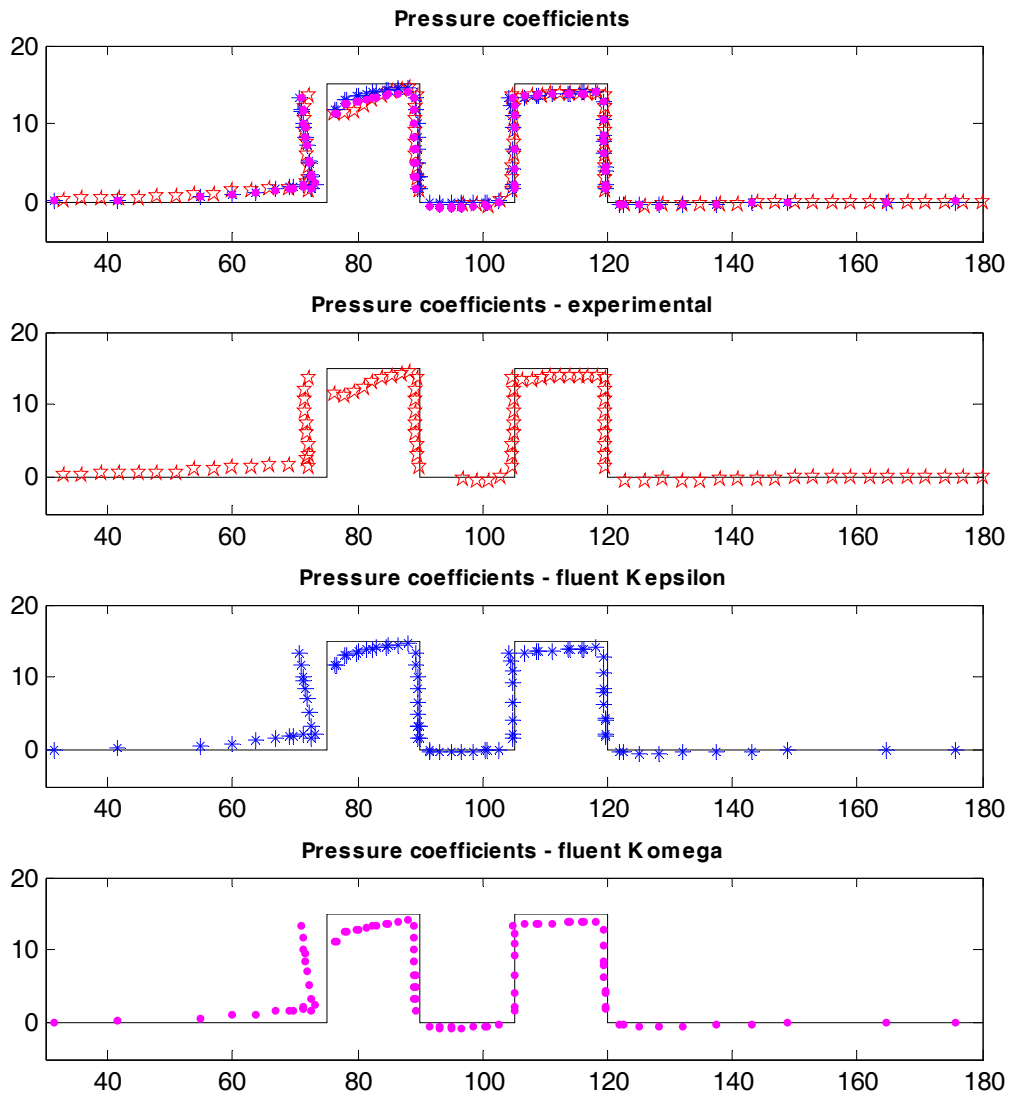


Figure 21 : model 3, pressure coefficients

5.3 Comments

In the first part of this study CFD simulations made with two turbulence models are compared to a physical model tested in wind tunnel trough optical velocimetry. The obtained information indicate the behavior of the flux in the region between the two. In each of the tree configuration tested is possible to see a good agreement of the results. The vortex structures of CFD are very similar to the ones measured, both in terms of vortex position and dimension; there is also correspondence in the signs of the angular velocity. A better accord is founded in the configurations number 2 and 3, while in the configuration number 1 the numerical model produces a lower vortex than the physical one.

An second validation of the numerical simulation has been made through the value of the static pressure measured in the middle plane on the walls of the building and on the floor. The formation of zones of overpressure and underpressure in a built area play a significant role in

the development of diffusion-related phenomena. For this reason the evaluation of the correct size of these zones is very important. In the cases investigated in the present study, both, in the numerical and in the physical model, the main features of the pressure contour are:

- zone of overpressure in front the building upwind;
- flow separation on the top of the building upwind;
- underpressure in the back side of the first building in configuration 2 and 3, overpressure in configuration 1;
- flow separation on the top of the building downwind;
- zone of underpressure behind the building downwind.

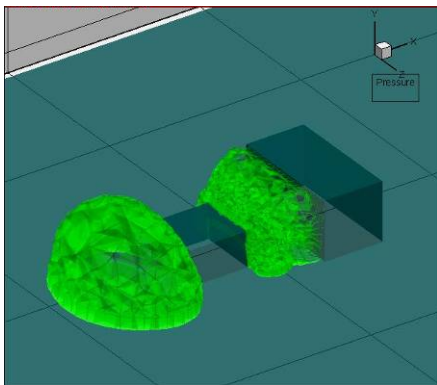


Figure 22 : Overpressure 'bubbles'

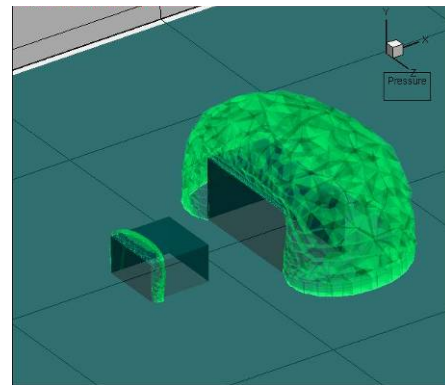


Figure 23 : Underpressure 'bubbles'

In order to compare the different results the dimensional characteristics of these structures have been quantified. All the configuration tested appear to be similar.

In the following tables (Tab.1, 2 and 3) some dimensional parameters, taken from the flow visualization, are reported, such as the coordinates of the center of the vortex and the stagnation point in the front (upwind) side of the second building. From the pressure measurements other two parameters are individuated: the total length of the overpressure zone in front of the building upwind and the total length of the zone of negative pressure behind the second building. The result that better agree with the experimental value is evidenced.

MODEL 1	<i>standard κ-ϵ</i>	<i>standard κ-ω</i>	Wind tunnel
Coordinates of the center of the vortex between the two buildings	(93.41, 2.32)	(93.44, 2.33)	(96.50, 3.38)
Coordinates of stagnation point in the front hedge of the second building	(105, 9.95)	(105, 10.12)	(105, 9.90)
length of overpressure zone in front of the first building [mt]	20	20	25
length of underpressure zone behind the second building [mt]	30	30	30

Table 1 : Some dimensional characteristics of the flow structures in the configuration number 1

MODEL 2	<i>standard κ-ϵ</i>	<i>standard κ-ω</i>	Wind tunnel
center of the vortex	(96.35, 6.17)	(96.17, 6.68)	(96.11, 5.45)
stagnation point in the second building	(105, 12.22)	(105, 12.58)	(105, 12.11)
length of overpressure zone in front of the first building [mt]	30	30	20
length of underpressure zone behind the second building [mt]	25	25	25

Table 2 : Some dimensional characteristics of the flow structures in the configuration number 2

MODEL 3	<i>standard κ-ϵ</i>	<i>standard κ-ω</i>	Wind tunnel
center of the vortex	(99.45, 11.18)	(96.72, 11.81)	(96.00, 10.32)
stagnation point in the second building	(105, 13.40)	(105, 14.00)	(105, 12.50)
length of overpressure zone in front of the first building [mt]	45	45	45
length of underpressure zone behind the second building [mt]	25	25	25

Table 3 : Some dimensional characteristics of the flow structures in the configuration number 3

6 CONCLUSION

The purpose of this study is to develop a three-dimensional CFD model able to simulate the flow field in a typical urban context, such as a group of buildings.

The objective is to use the optimal experimental set-up as test-case for further investigations about concentration of gaseous and particulate pollutants dispersion through an advection-diffusion model.

The sensitivity of the results to some computational parameters such as the grid definition (extension of the domain, hybrid structure, resolution, ...) and the turbulence model are evaluated.

The domain dimensions chosen for the numerical simulation results to be appropriate; the distance of the outflow boundary from the buildings is sufficient to avoid backflow reentering the domain and the total height is been raised to 100 m (6.6 x height of the buildings) in order to avoid distortions of the flow. Trough a grid refinement procedure the independence of the solution to the grid choice has been verified.

This formulation of the dissipation rate used in the CFD simulation does not guarantee the maintenance of the turbulent kinetic energy trough the entire domain.

The main characteristics of the flow structure established in the numerical simulation seem to be very similar to the ones individuated through the wind tunnel experimental campaign and so another epsilon formulation will be tested hereafter.

The most appreciable differences are an underestimation (by the numerical result) of the negative pressure on the top of the building downwind in configurations 1 and 2 (Fig.17 and 18) and the height of the vortex between the two buildings in configuration 1 (Fig. 8, 9 and 10).

The κ - ε turbulence model seem to conduct to a better agreement than the κ - ω model, even if this behavior is not marked. Little differences can not be noticed due to the resolution of the measuring arrangement (the cross-correlation areas used in the PIV elaboration are of 32x32 pixel, equal to about 0.66 cm in the model and 1 m in full scale; the pressure tabs were positioned at a mutual distance of 1 cm on the buildings walls and 2 cm on the floor, in full scale equal to 1.5 m and 3 m respectively).

ACKNOWLEDGEMENTS

This study has been carried on with the collaboration of ARPAT (Regional Environmental Protection Agency of Tuscany) for the selection of the domain and of the Energetic Department S. Stecco of the University of Florence for the support on computational practice; the experimental campaign has been conduced in the Boundary Layer Wind Tunnel of CRIACIV in Prato, Italy.

REFERENCES

- [1] COST Action 732, 2005-2009. Quality assurance and improvement of micro-scale meteorological models. (<http://www.mi.uni-hamburg.de/Home.484.0.html>).
- [2] Di Sabatino, S., Buccolieri, R., Pulvirenti, B., Britter, R., 2007b. Simulation of pollutant dispersion within idealised urban-type geometries with CFD and integral models. *Atmospheric Environment* 41 (2007), pp. 8316–8329.
- [3] Di Sabatino, S., Buccolieri, R., Pulvirenti, B., Britter, R., 2007b. Flow and pollutant dispersion modelling in street canyons using Fluent and ADMS-Urban. *Environmental Modeling & Assessment*, doi:10.1007/s10666-007-9106-6.
- [4] D. Hargreaves and N. Wright, 2006. On the use of k-e model in commercial CFD software to model the neutral atmospheric boundary layer. *Journal of Wind Engineering and Industrial Aerodynamics*, 95 (2005), pp 355-369.
- [5] N. Holmes 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 (2006), pp. 5902–5928.

- [6] K.J. Kim and J.J. Baik, 2004. A numerical study of the effects of ambient wind direction on flow and dispersion in urban street canyons using the RNG k-e turbulence model. *Atmospheric Environment* 38 (2004), pp. 3039–3048.
- [7] A. Riddle, D. Carruthers, A. Sharpe, C. McHugh and J. Stocker, Comparisons between FLUENT and ADMS for atmospheric dispersion modelling, *Atmospheric Environment* 38 (2004), pp. 1029–1038.
- [8] E. Simiu, R. Scanlan, 1996. Wind effect on structures, T.rd edition. Wiley-Interscience Publications.
- [9] D. C. Wilcox, 1993. Turbulence modeling for CFD. DCW Industries, Inc. La Canada, California.