BBAA VI International Colloquium on:
Bluff Bodies Aerodynamics & Applications
Milano, Italy, July, 20-24 2008

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, pollutant dispersion, modeling

1 INTRODUCTION

Urban planning toward air quality and related health risks monitoring is currently increasing and actually needs trustful diagnostic and prediction tools to assess ambient pollution.
Until yet, the most common way of dealing this issue has been through operational non-CFD models, 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 this cases.
The main purpose of this study is to find an optimal test case for a CFD simulation in order to have a standard numerical tool of prevision to be used for similar cases in an urban context. The goodness of this set-up and its sensitivity to some parameters will be estimated through an experimental campaign.
In this paper the flow field around a group of buildings has been evaluated using the commercial CFD code FLUENT; the results have been validated through wind tunnel measurement.
The κ–ε turbulence model has been used for CFD simulations, widely used in wind engineering and in near-field dispersion models (Hagreaves and Wright 2006, Kim and Bike 2004, Launder and Spalding 1974). The experimental data have been achieved through flow visualization and flow velocity measurement (PIV techniques) in an experimental campaign conducted in the Boundary Layer Wind Tunnel of CRIACIV, (Florence, Italy). Both the CFD simulation and the wind tunnel tests are the first step for a further investigation including dispersion analyses and concentration measurement for the same study case.

2 CFD MODEL

The choice of the test configuration comes from a typical situation of urban environment, where a small building is set upwind respect to a higher one. The main concern is to evaluate the dispersion pattern of emissions from the small building and affected by the higher one.
The dimensions of the 'blocks' of the model are realistic for a urban environment in Florence, in which buildings range from 1 floor (3m high) up to 5 m floors (15 m high). The configurations of the model have been set up by varying the small building height: 3m, 9m and 15m.

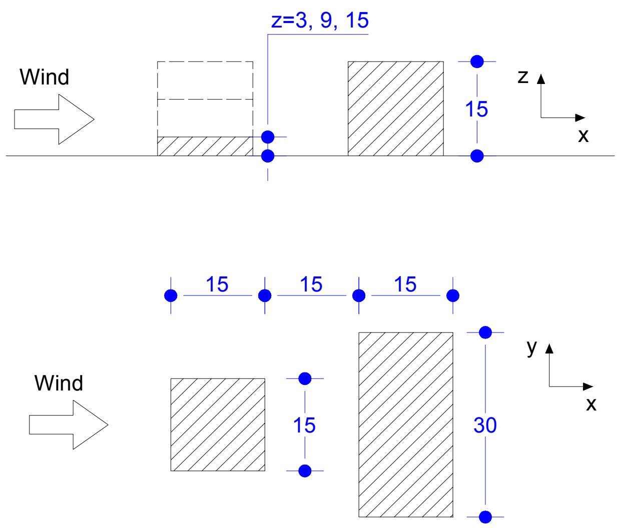

Figure 1: Sketch of the model, plane x-z and x-y.

The flow around the buildings in neutral stability conditions has been carefully investigated (stagnation point, separation zones, evolution of the wake) with different inlet wind conditions.

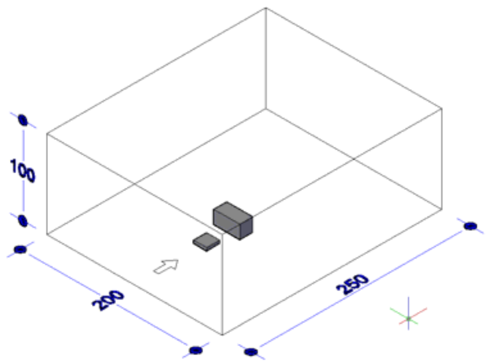

Figure 2: Domain of the CFD model.

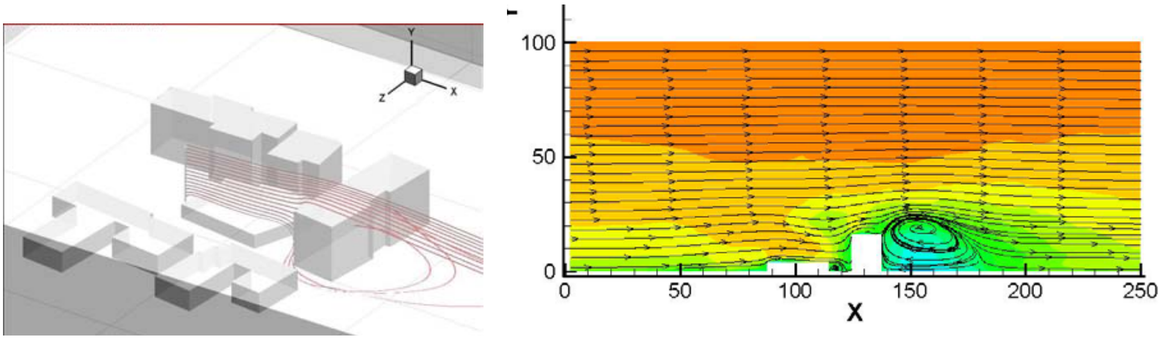

Figure 3:Example of flow visualization in urban context.

3 WIND TUNNEL TESTS

3.1 Description of the model

The same model of the CFD simulation (Fig.1) has been reproduced in small scale 1:100. The model has 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 roughness elements at the floor.

3.2 Measuring set-up

The characterization of the incoming wind profile has been revealed trough velocity measurement with a hot wire anemometer at many heights in the middle plane of the test section.
Measurements of the flow field in different planes have been made trough Particle Image Velocimetry (PIV) technique and flow visualization of the flow trough a seeding has been made as well.

4 CONCLUSIONS

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 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 experimental set-up will be used as test-case for further investigations about concentration of gaseous and particulate pollutants dispersion through an advection-diffusion model (Di Sabatino 2007, Riddle 2003).

ACKNOWLEDGEMENTS

This study has been carried on with the collaboration of ARPAT (Agency for environment) and of the Energetic Department S. Stecco of the University of Florence; 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 dispertion 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.