FLOW AND POLLUTANT DISPERSION IN URBAN ARRAYS FOR THE STANDARDIZATION OF CFD MODELLING PRACTISE

Riccardo Buccolieri and Silvana Di Sabatino
Dipartimento di Scienza dei Materiali, University of Lecce, Italy

ABSTRACT
The study of the effect of obstacles on flow and dispersion in the atmospheric boundary layer is one of the most important topics in atmospheric dispersion research. Computational Fluid Dynamics (CFD) methods are increasingly used to predict concentration fields near buildings in an operational context, but extensive validations are still needed. In this paper, we analyse the effect on flow and dispersion due to the presence of a building array placed in the atmospheric boundary layer. The main aim is that of improving our current knowledge of the application of CFD methods to new case studies and contributing to the standardisation of CFD modelling practise in a wider context. Our analysis is based on the experience gained by the comparison of CFD numerical simulations, obtained with the commercial CFD model FLUENT, with wind tunnel data sets from the MUST experiment.

INTRODUCTION
Flow patterns around buildings have a strong influence on pollutant dispersion from sources placed within the urban area. The prediction of ground-level pollutant concentrations is important for the assessment of the impact of existing sources on people health and the environment. Traditionally, information about flow and pollutant concentrations has been obtained using field and wind tunnel experiments. In the MUST (Mock Urban Setting Test) (*Yee, E. and C.A Biltoft*, 2004) experiment, a large outdoor field study which has been reproduced in wind tunnel, it was attempted to simulate an urban boundary layer by the construction of a regular array of shipping containers in near-neutral atmospheric conditions. Recently, Computational Fluid Dynamics (CFD) has become an attractive tool to predict concentration fields near buildings, but at present there are not yet best-practice methodologies for using CFD as an operational tool. In this paper, we simulate the MUST experiment using the CFD code FLUENT (*FLUENT 6.2*, 2005). CFD simulations are performed using both the $k\text{–}\varepsilon$ and the Reynolds-Stress turbulence models. The present work is part of our current research within the *COST 732 Action* (2005-2009), devoted to the study of the effect of obstacles on flow and dispersion in the real urban environment and to the standardization of CFD modelling practise for atmospheric applications. The work may be considered a verification of the efficiency of the Protocol and the Best Practice Guidelines (*BPG*) of the COST 732 Action and an assessment of their applicability. Here we present and discuss CFD simulations results for the mean velocity components and for turbulent kinetic energy using 0° and -45° approaching flow conditions. Pollutant dispersion results within the same array is presented and discussed in *Di Sabatino, S. and R. Buccolieri* (2007).

METHODOLOGY
Description of wind tunnel experiments (MUST)
The wind tunnel data set used in this paper contains flow and dispersion data measured within an idealized urban roughness. The wind tunnel experimental setup originates from the Mock Urban Setting Test (MUST), an extensive field test carried out on a test site of the US Army in the Great Basin Desert in 2001. 120 standard size shipping containers were set up in a nearly regular array of 10 by 12 obstacles. The wind tunnel measurements within a scaled

model (1:75) of that configuration were carried out at the University of Hamburg (*Bezpalcova, K. and F. Harms*, 2005). We focus only on two main wind directions (0° and -45°) which correspond to those cases selected within the COST 732 Action.

CFD modelling: geometry and grid

Building dimensions within the MUST array are given by points, so the geometry is built by entering in GAMBIT all the corners of each building. The geometry modelled in GAMBIT is the exact representation of that used in full scale experiments. The grid used is structured (hexahedral cells) with refinement (the expansion rate between two consecutive cells is below 1.3 in regions of high gradients). Several tests on the influence of computational domain and grid size are performed to ensure the independence of the solution from the domain and the grid (see next paragraph). The grid was imported in FLUENT and scaled as performed in wind tunnel experiments (1:75).

CFD modelling: flow setup

Simulations are carried out by considering a neutral boundary layer. The standard k-ε and the Reynolds Stresses models are used. Based on wind tunnel experiments the inlet wind speed is assumed to follow a logarithmic law profile with height z:

$$\frac{U(z)}{U_{ref}} = \frac{u_*}{\kappa}\ln\left(\frac{z}{z_0}\right) \qquad (1)$$

where $U(z)$ is the average wind speed at the height z above the ground, U_{ref}=5.5 m/s is the reference wind speed (undisturbed flow), z_0=2.27^{-04} m (only upwind ground) is the roughness height, u_*=0.36 m/s is the friction velocity and κ the von Karman's constant. Turbulent kinetic energy and dissipation rate profiles are specified as follows (*Hargreaves, D. and N. Wright*, 2006):

$$k = \frac{u_*^{\,2}}{\sqrt{C_\mu}} \text{ and } \varepsilon = \frac{u_*^{\,3}}{\kappa(\mathrm{z}+\mathrm{z}_0)} \qquad (2)$$

where C_μ=0.09 is a coefficient used to define the eddy viscosity in k-ε models. The remaining boundary conditions (surface roughness representation, symmetry conditions etc.) are those specified in *Di Sabatino, S. et al.* (in press, 2007). The convergence criterion is 10^{-6} for all variables.

RESULTS AND DISCUSSIONS

FLUENT simulation results are compared with MUST wind tunnel data focusing on the mean velocity components and the turbulent kinetic energy (TKE). To assess model performance several statistical methods have been used. However, here we only report on results from the Hit Rate validation test q (*Schlünzen, K.H. et al.*, 2004). This has been calculated using a fractional deviation RD=0.25 and an absolute deviation W=0.06, recalling that q>66% is requested for the comparison with wind tunnel data.

Influence of domain size

The *BPG* suggests specific values for the dimensions of the computational domain when a single building is modelled. To test the optimum domain size in presence of a building array a domain independence test is performed. In particular we test the influence on the flow predictions of several geometric features. These are: the distance from the inlet plane (L) to the first buildings and the distance above the ground (Y). The grid is chosen based on the suggestion of at least 10 cells per cube root of the building volume and 10 cells per building separations given in the *BPG*. Table 1 summarizes the dimensions used for the different computational domains. Domain β and ζ give similar results, therefore β is used in the remaining simulations as it gives the smallest domain size (Fig. 1).

Table 1. Distance from the inlet plane (L) to the first buildings and distance above the ground (Y) used for testing the influence of computational domain. H_{max}=3.51m (full scale) is the height of the tallest building.

Domain	Y	L (fixed)	Domain	L	Y (fixed)
α	$3H_{max}$		δ	$3H_{max}$	
β	$5H_{max}$	$5H_{max}$	β	$5H_{max}$	$5H_{max}$
γ	$7H_{max}$		ζ	$7H_{max}$	

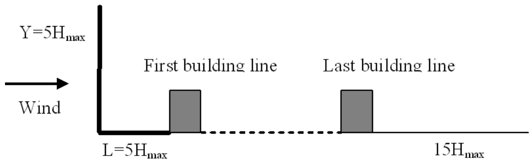

Fig. 1; Schematic view of the extent characteristics of the computational domain used.

Influence of grid size

Before choosing the grid distribution, the influence on the predictions of the choice of grid size is determined using the *k-ε* model (Table 2). The *BPG* gives some suggestions about the computational grid size, even if the grid resolution is a highly problem dependent. The suggestion of at least 10 cells per cube root of the building volume and 10 cells per building separations is considered as initial minimum grid resolution (coarser grid). The medium and the finer grids give similar results and the medium is used in the remaining simulations. The number of cells is about 1.300.000 for all cases.

Table 2. Characteristics of grids used for testing the influence of grid size. δ_{xmin}, δ_{ymin} and δ_{zmin} refer to the size (normalised with H_{max}) of the smallest grid spacings in the x, y and z directions, respectively.

Grid	δ_{xmin}	δ_{ymin}	δ_{zmin}
Coarser		0.28	0.11
Medium			0.05
Finer		0.14	0.02

Comparison of FLUENT results with experimental data

Flow results are plotted inside street canyons and intersections at the beginning, middle and end of the building array. For the 0° approaching flow case, vertical profiles (Fig. 2) and the Hit rate test (Table 3) show that there is a good agreement at intersections, even if we observe a general light overestimation inside and over the canyons. In general, the Reynolds Stress Model gives similar results than the *k-ε* model. For the -45° approaching flow case, vertical profiles (Fig. 3) and the Hit rate test (Table 4) show better comparison results both at intersections and in street canyons. The Hit rate test is in fact fulfilled in all positions.

Table 3. 0° approaching flow case: Hit rate test.

0° approaching flow case		beginning		middle		end	
		SC	INTERSECTION	SC	INTERSECTION	SC	INTERSECTION
u/U_{ref}	*k-ε*	0.68	1	0.41	0.85	0.44	1
	RSM	0.39	1	0.34	1	0.63	1
TKE/U^2_{ref}	*k-ε*	0.56	1	1	1	1	1
	RSM	0.52	1	1	1	1	1

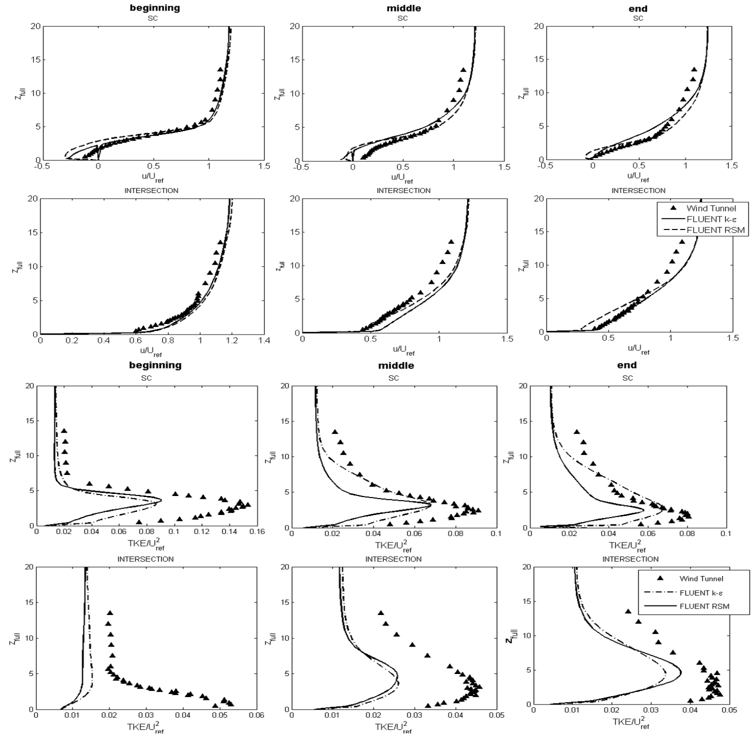

Fig. 2; 0° approaching flow case: vertical profiles of u/U_{ref} (top) and TKE/U^2_{ref} (bottom).

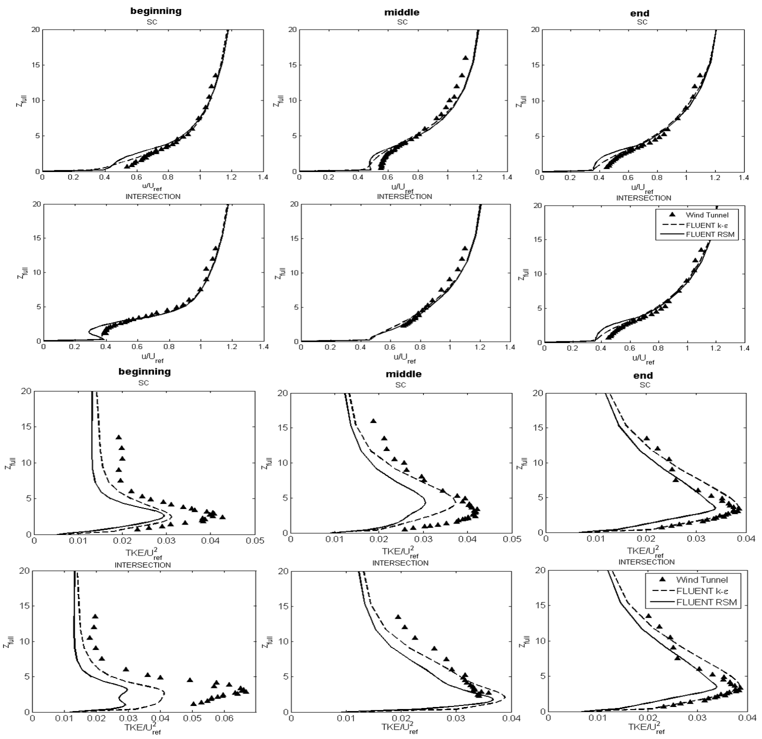

Fig. 3; -45° approaching flow case: vertical profiles of u/U_{ref} (top) and TKE/U^2_{ref} (bottom).

Table 4. -45° approaching flow case: Hit rate test

-45° approaching flow case		beginning		middle		End	
		SC	INTERSECTION	SC	INTERSECTION	SC	INTERSECTION
u/U_{ref}	*k-ε*	0.95	1	1	1	1	1
	RSM	1	0.84	1	1	1	1
TKE/U^2_{ref}	*k-ε*	1	1	1	1	1	1
	RSM	1	1	1	1	1	1

CONCLUSIONS

We show that, with the methodology set up in our previous works, flow within complex geometries can be modelled with the CFD code FLUENT with a certain degree of reliability. This CFD modelling practise allow us to give suggestions to improve the COST 732 Action documents. In fact, the novel aspects of this paper are the validation of the CFD code in non-cubical buildings and the verification of the validity of general simulation criteria found for the simplest configurations in more complex conditions. Similar criteria for grid refinement and parameters used for the single building and street canyons cases can be adopted for complex building arrays. For example, a grid independence solution can be obtained with a grid dimension close to the building equal to 0.05 H. About the computational domain size 20-25 cells per cube root of the building volume and 15-20 cells per building separations could be an adequate choice for regular building arrays.

REFERENCES

Bezpalcova, K. and F. Harms, 2005: EWTL Data Report / Part I: Summarized Test Description Mock Urban Setting Test. *Environmental Wind Tunnel Laboratory, Center for Marine and Atmospheric Research, University of Hamburg*

Di Sabatino, S. and R. Buccolieri, 2007: MUST experiment simulations using CFD and integral models. *Proc. 11th conference on Harmonisation within Atmospheric Dispersion Modelling for Regulatory Purposes, Cambridge, UK, July 2-5.*

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

Di Sabatino, S., R. Buccolieri, B. Pulvirenti and R. Britter, 2007: Application and validation of FLUENT flow and dispersion modelling within complex geometries. In: Borrego C. and E. Renner (Eds), Air Pollution Modeling and its Application XVIII, ISSN: 1474-8177/DOI:10.1016/S1474-8177(07)06011-1.

Di Sabatino, S., R. Buccolieri, B. Pulvirenti and R. Bitter, in press: Flow and pollutant dispersion modelling in street canyons using Fluent and ADMS-Urban. *Environmental Modelling & Assessment.*

FLUENT 6.2, 2005: User's Manual. *http://www.fluent.com.*

Hargreaves, D. and N. Wright, 2006: On the use of the *k-ε* model in commercial CFD software to model the neutral atmospheric boundary layer. *J. Wind Eng. Ind. Aerodyn.*, **95**, 355–369.

Schlünzen, K.H., W. Bächlin, H. Brünger, J. Eichhorn, D. Grawe, R. Schenk and C. Winkler, 2004: An evaluation guideline for the prognostic microscale wind field models. *Proc. 9th International Conference on Harmonisation within Atmospheric Dispersion Modelling for Regulatory Purposes, Garmisch-Partenkirchen (Germany).*

Yee E. and C.A. Biltoft, 2004: Concentration fluctuation measurements in a plume dispersing through a regular array of obstacles. *Boundary Layer Meteorology*, **111**, 363-415.