Intercomparison of Numerical Urban Dispersion Models – Part I: Street Canyon and Single Building Configurations

P. Sahm[1*], P. Louka[2], M. Ketzel[3], E. Guilloteau[4] and J.-F. Sini[2]
[1] Laboratory of Heat Transfer and Environmental Engineering, Aristotle University Thessaloniki, Greece
[2] Fluid Mechanics Laboratory, Ecole Centrale de Nantes, Nantes, France
[3] Department of Atmospheric Environment, National Environmental Research Institute, Roskilde, Denmark
[4] Institute of Hydromechanics, University of Karlsruhe, Karlsruhe, Germany

1. Introduction

Traffic pollution in urban areas has, in the last decades, become a major hazard to public health. The aggregation of human activities, especially in areas of insufficient ventilation, has often led to pollutant concentration levels much higher than the limits set by the World Health Organisation. The building aggregates, placed within the atmospheric boundary layer, act as artificial obstacles to the wind flow and cause stagnant conditions in the city, even for relatively high ambient wind conditions. A typical configuration is the so-called street canyon, formed along a street in densely built urban areas.

The research topic for the TRAPOS network concerns improvement and optimisation of the methods that are used for modelling of traffic pollution in streets. Based on evaluation of presently available models and study of results from field and laboratory experiments, the major deficits in the description of important processes have been identified and then addressed. The study included measurements in streets, wind tunnel experiments, and advanced numerical modelling and data evaluation. The main objectives of the project are to improve the performance of models in the case of critical meteorological conditions, to improve their ability to deal with different street architectures, to improve description of the chemical and physical conversion processes and treatment of new, not previously addressed pollutants.

The numerical models used within the TRAPOS network comprise five advanced CFD models, i.e. CFX-TASCflow (Raw et al., 1989), CHENSI (Sini et al., 1996), CHENSI-2 (Guilloteau, 1999), MIMO (Ehrhard et al., 2000) and MISKAM (Eichhorn, 1989), for the numerical simulation of the three-dimensional fluid dynamics and the dispersion of pollutants in the microscale. A detailed description of the models is available on the websites of the TRAPOS network (http://www.dmu.dk/AtmosphericEnvironment/trapos/data_and_models.htm, http://www.dmu.dk/AtmosphericEnvironment/trapos/cfd-wg.htm).

Through close co-operation between the network teams an extensive CFD model evaluation study was organised. Several test cases have been defined, ranging from a single cavity case and a simple 3D case, to real case exercises. For all cases either very comprehensive field data sets or wind tunnel measurements were available. This paper, which is the first in a sequence of two, is focussed on the intercomparison of the models in the cases of a single cavity and a surface mounted cube. The intercomparison on a real case (i.e. a street canyon in Hannover, Germany) is described in the accompanying part II paper (Ketzel et al., 2001).

2. Single cavity case

The single cavity case was defined as the simplest two-dimensional case to investigate the performance of the codes in reproducing the flow field between buildings. The experimental database for this case was established in the wind tunnel of the University of Surrey. This database was developed for different cavity dimensions and the experiments aimed at assessing the effect of the cavity dimensions on the transformation of the flow from the one regime to the other and consequently on the dispersion of pollutants within a 'real' street. The first experimental case chosen to be studied by the numerical codes was a single cavity with aspect ratio, i.e. width of cavity, W, over its depth, H, W/H, equal to 2 (Figure 1). Vertical profiles of the mean wind field (u and w components), and the turbulent kinetic energy, k, were measured upstream of the cavity and were specified as the input data for the models. Similar measurements were also performed within and above the cavity at positions $x/W \approx 0.1$, 0.3, 0.5, 0.7, and 0.9. All five models used the same domain and grid sizes.

The flow pattern observed within the cavity in the wind tunnel was generally reproduced by the models. The flow within the cavity is dominated by a main re-circulation, while a secondary vortex rotating in the opposite direction is present at the leeward side of the cavity close to the ground (Figure 1). The re-circulation predicted by the models is characterised by mean velocities ranging between –2m/s and 2m/s and the secondary vortex by lower velocities between -0.15m/s and 0.15m/s. The comparison between the numerical results and the measurements at the different positions within the cavity shows a fairly good agreement especially for the mean wind field. Figure 2 shows the vertical profile of the *u* component at *X/W*=0.1, 0.5, and 0.9. The models have captured both the shape of the profile and roughly the magnitude of the wind speed.

It is observed that close to the solid boundaries (building walls and ground) the models show the largest divergence in their predictions. The detailed examination of the source code showed that the origin of this difference is mainly due to the different implementation of the wall functions. It is generally observed that CHENSI and CFX-TASCflow predict the same velocity values close to the walls, while CHENSI-2 and MIMO following the same wall-function implementation calculate very similar velocities. Due to the implementation of the advection scheme and boundary conditions on solid surfaces, MISKAM is mainly dedicated in simulating real-site flows therefore is probably less accurate in estimating small-scale flow patterns. The intercomparison showed that at very low wind conditions, e.g. at *X/W*=0.1, the effect of the different wall-function implementation by the codes on the calculated velocities is small. However, differences are still observed in the *k*-profiles (not shown here).

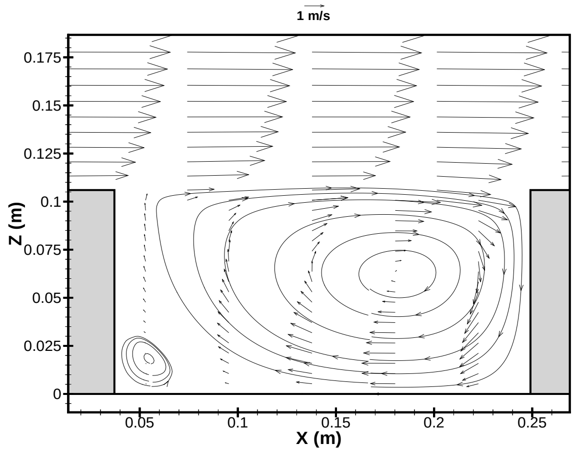

Figure 1. The flow field within and above the cavity as it was reproduced by CHENSI.

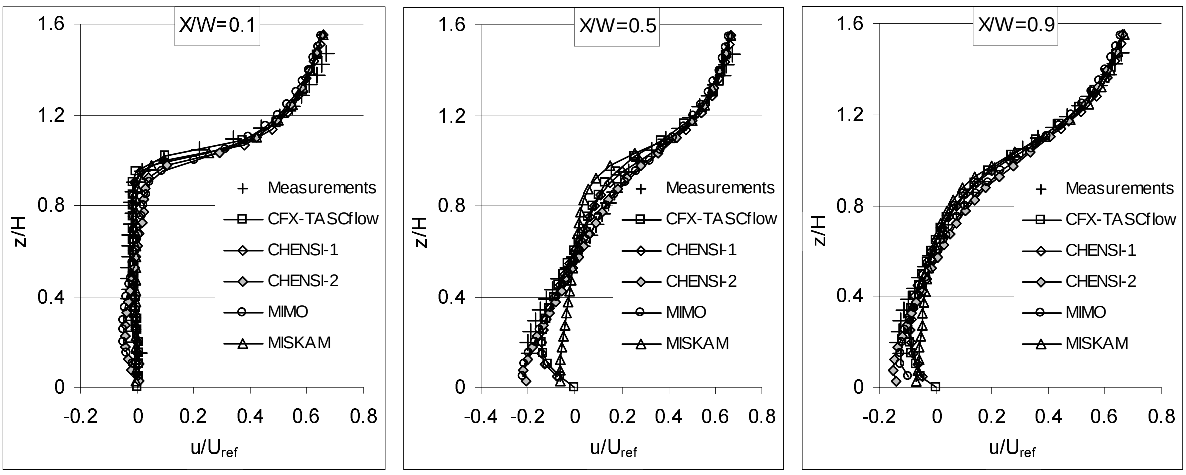

Figure 2. Comparison of the profile of the *u*-component normalised with the free-stream velocity (U_{ref} = 8 m/s) observed in the wind tunnel with that predicted by the numerical models at *X/W*=0.1, 0.5, and 0.9.

3. Flow around a wall-mounted cube

The wall-mounted cube case was defined as the simplest three-dimensional case to investigate the performance of the codes in reproducing the flow field around a building. The experiment was carried out in the BLASIUS wind tunnel at the Meteorological Institute of Hamburg University. A wooden cube (125mm×125mm×125mm in model scale) was used to simulate an idealised model building of 25m height (full scale). The cube was mounted in the centre of the wind tunnel test section.

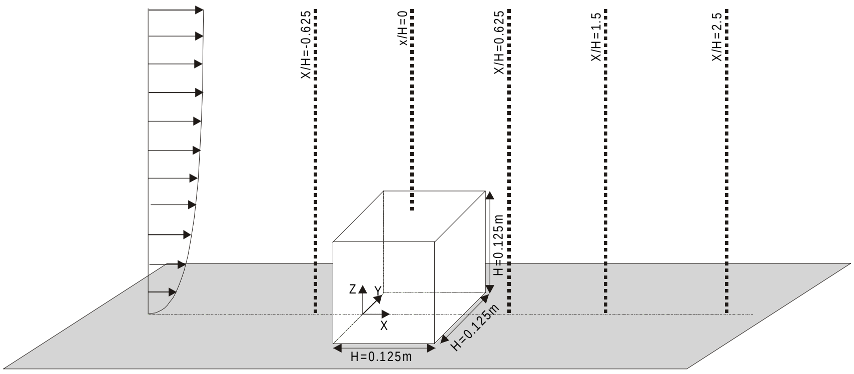

Figure 3. Experimental set-up of the wall-mounted cube; locations of the measuring positions in the centre plane.

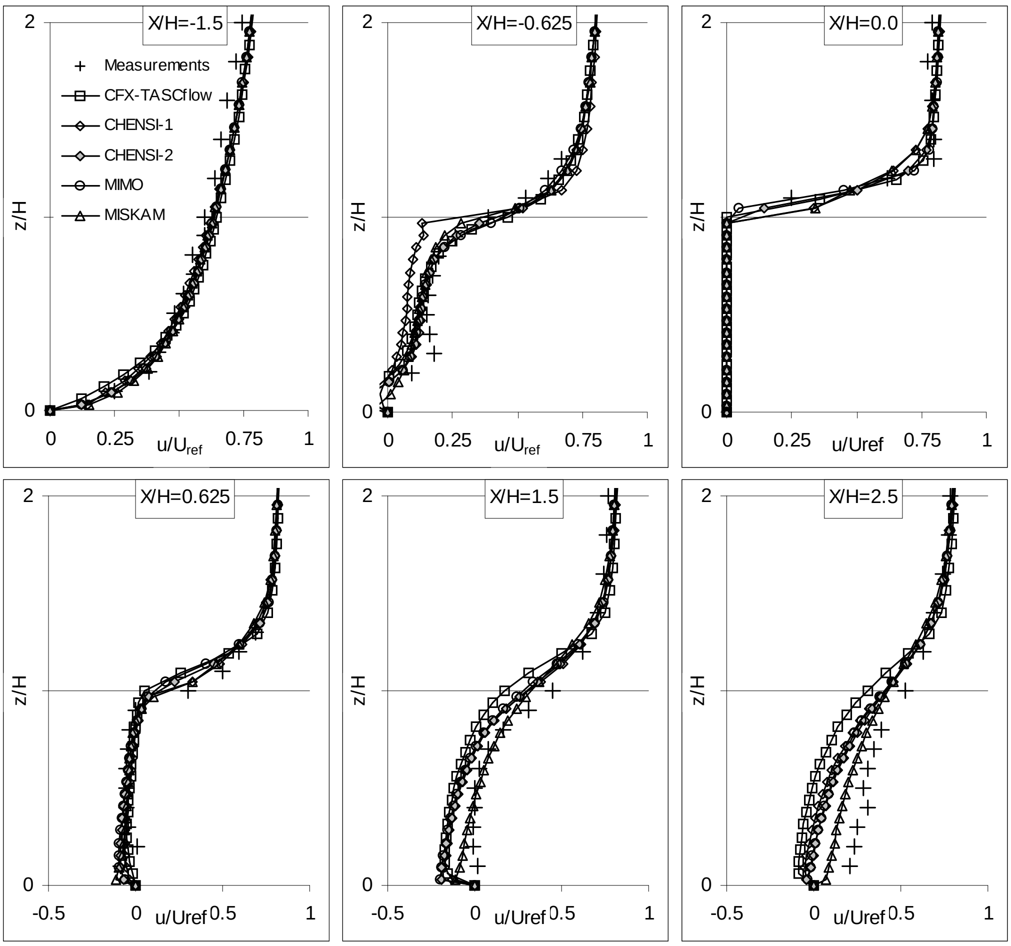

Figure 4. Comparison of the dimensionless profile of the *u*-component normalised with the free-stream velocity (U_{ref} = 6 m/s) observed in the wind tunnel with that predicted by the numerical models in the centre plane of the flow at various distances.

The vertical profile of the mean wind field was measured upstream of the obstacle (i.e. at *x/H*=-8) and was used to derive u* and z_0 and subsequently the logarithmic law was applied to obtain the vertical profile of u at inflow. For the turbulent kinetic energy k and the dissipation rate ε the approximations $k=u^{*2}/\sqrt{(c_\mu)}$ with $c_\mu=0.09$ and $\varepsilon=u^{*3}/(\kappa\cdot z)$ were used to specify the input data for the models. All three models used the same domain and grid sizes.

Figure 4 shows the vertical profile of the u-component at inflow and at different positions (*X/H*=-1.5, -0.625, 0, 0.625, 1.5 and 2.5) in the centre plane of the flow. The agreement between measured and computed data at *X/H*=-1.5 is excellent, hence it can be argued that the computations have managed to simulate the exact experimental conditions prevailing upwind of the obstacle. The comparison between the numerical results and the measurements at the different positions *X/H* shows a good agreement for locations above the cube (e.g. at dimensionless heights *Z/H*>1). In agreement with observations, all models predict large gradients of the u-component within the re-circulation area (*X/H*=0.625). The re-circulation predicted by the models is characterised by mean velocities ranging between –1 m/s and –1.5 m/s. Close to the observed reattachment point in the centre plane (*X/H*=1.5) all models compute a negative velocity close to the surface indicating that this position is predicted to be still far inside the re-circulation area, thus the models overestimate the reattachment length x_R.

Except to CHENSI predictions, all models perform similar upstream the obstacle; the method of implementing the wall functions in this model most probably being the main reason for the deviation. Largest differences among the model predictions can be found in the re-circulation zone: Compared to the other models, MISKAM seems not to capture in full the re-circulation, while CFX-TASCflow results show the most extended re-circulation zone. Throughout the whole test case, CHENSI-2 and MIMO predictions lead to similar results.

4. Conclusions

Improvement and optimisation of the methods used in practical application of traffic pollution models for air quality impact studies is one of the final goals of the TRAPOS network. Within the network CFD model intercomparison exercises were organised for model evaluation. The results of the model intercomparison indicate that the applied models show, in general, a reasonable skill in predicting the main features of the flow and dispersion conditions. As one main aim of this intercomparison was to demonstrate the expected level of agreement of similar codes for these generally simple study cases, it is suggested that the implementation of the wall-function has small effects on velocity calculations at locations of low wind speeds.

5. Acknowledgements

The presented model intercomparison was carried out in the frame of the research network TRAPOS which operates in the framework of the European Commission Training and Mobility of Researchers Programme. The authors wish to acknowledge the provision of input data for the model comparison by Drs Savory and Kovar-Panskus, University of Surrey and Drs Leitl and Schatzmann, University of Hamburg. Computer support was given to the French team by the Scientific Council of Institut de Développement et de Recherche pour l'Informatique Scientifique -IDRIS- (CNRS), Orsay, France.

References

Ehrhard J., Khatib I.A., Winkler C., Kunz R. Moussiopoulos N. and Ernst G. (2000) The microscale model MIMO: development and assessment, *J. of Wind Engineering and Industrial Aerodynamics*, **85**, 163-176.

Eichhorn, J. (1989) Entwicklung und Anwendung eines dreidimensionalen mikroskaligen Stadtklima-Modells. Dissertation, Universität Mainz, Germany.

Guilloteau E. (1999) Modélisation des sols urbains pour les simulations de l' atmosphère aux échelles sub-méso, PhD thesis, University of Nantes - Ecole Centrale de Nantes, France.

Ketzel, M., Louka, P., Sahm, P., Guilloteau, E.., Sini, J.-F. and Moussiopoulos, N. (2001), Intercomparison of Numerical Urban Dispersion Models – Part II: Street Canyon in Hannover, Germany, 3rd International Conference on Urban Air Quality, 19-23 March, 2001, Loutraki, Greece.

Raw, M.J., Galpin, P.F. and Hutchinson, B.R. (1989) A collocated finite-volume method for solving the Navier-Stokes equations for incompressible and compressible flows in Turbomachinery: Results and applications, *Canadian Aeronautics and Space Journal* **35**, 189-196.

Sini, J.-F., Anquetin S. and Mestayer P.G. (1996) Pollutant dispersion and thermal effects in urban street canyons, *Atmos. Environ.* **30**, 2659-2677.