CFD INTERCOMPARISON EXERCISE WITHIN TRAPOS EUROPEAN RESEARCH NETWORK

Louka P.[1,2], Ketzel M.[3], Sahm P.[2], Guilloteau E.[4], Moussiopoulos N.[2], Sini J.-F.[1], Mestayer P.G.[1], Berkowicz R.[3]

[1] Laboratoire de Mécanique des Fluides CNRS - Ecole Centrale de Nantes, 1 Rue de la Noë, P 92101, 44321 Nantes Cedex 3, FRANCE
[2] Laboratory of Heat Transfer and Environmental Engineering, Aristotle University of Thessaloniki, PO Box 483, 54006 Thessaloniki, GREECE
[3] Department of Atmospheric Environment, National Environmental Research Institute, Roskilde, DENMARK
[4] Institute of Hydromechanics, University of Karlsruhe, Karlsruhe, GERMANY

ABSTRACT

The results of a Computation Fluid Dynamics (CFD) intercomparison exercise that was performed by the working group on CFD modelling of the European research network of TRAPOS are presented. Several numerical models (CFX-TASCflow, CHENSI, CHENSI-2, MIMO, MISKAM) employing the widely used 'standard k-ε -model' were applied to well defined test cases and their results are compared. The cases treated were a two-dimensional single cavity, a single cube and a full-scale street canyon. Comparison of the model predictions with the available data is performed. The simpler study cases (single cavity and cube) demonstrate the level of agreement expected between similar codes and aims at improving and further developing the models. The discussion on the full-scale street case aims at explaining the differences along with giving some recommendations to CFD model users on how to calculate such complex flows and give useful suggestions to experimentalists on the concentration measuring position in order to achieve a more representative picture of air pollution dispersion in a street.

1. INTRODUCTION

The European research network of TRAPOS (URL1) operated within the framework of Training and Mobility of Researchers programme of the European Commission. It was initiated in 1998 and was completed in May 2001. The main objectives of the network was to provide research training especially to young researchers and to contribute to scientific developments within the research topic of air pollution in streets. Several research working groups were operated within TRAPOS and here we shall present the results of a CFD intercomparison exercise that was performed by the CFD modelling working group (URL2).
Traffic pollution in urban areas has, in the last decades, become a major concern for public health, since emissions from non-traffic sources have been constantly reduced. 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. Microscale CFD models have become an efficient and common simulation tool for assessment and prediction of air quality in urban areas. The fundamental problem of CFD simulations lies on the physical difficulties of modelling the effects of turbulence as well as on the accuracy of the spatial discretisation of complex urban geometries, the numerical procedure applied, the boundary conditions and physical properties selected. Therefore, the proper validation of such models is a crucial prerequisite for its practical application.

2. METHODOLOGY

This TRAPOS CFD exercise was based on laboratory and field experimental data sets collected by TRAPOS members and aimed at (a) assessing and designating the differences appearing when CFD codes using the same turbulence model are applied to the same well defined cases, (b) improving the knowledge base for the model development and application, (c) illustrating the level of agreement that can be expected from CFD modelling in urban environments, and (d) bringing forward some recommendations for practical applications.

2.1. Models

The CFD codes applied in this exercise 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 flow field and the dispersion of pollutants in the microscale. These models employ the widely used 'standard k-ε -model' for the turbulence closure but different implementation of the boundary conditions and different numerical schemes are used. A detailed description of the codes can be found in the model inventory (see link in URL2). All five models used the same domain and grid sizes as well as the same inflow conditions specified by the available experimental data sets.

2.2. Field and laboratory data

The test cases chosen comprised a variety of two and three-dimensional configurations for which measurements from wind tunnel and field studies were available. In particular, the cases treated were a two-dimensional single cavity, a single cube and a real street canyon (Goettinger Strasse) in Hanover, Germany.
The experimental database for the single cavity case was established in the wind tunnel of the University of Surrey (Kovar-Panskus et al., 2001). This database was developed for different cavity dimensions and the experiments aimed at assessing the effect of the cavity aspect ratio, i.e. width of cavity, W, over its depth, H, W/H, on the airflow and consequently on the dispersion of pollutants within a 'real' street. The experimental case chosen to be studied by the numerical codes was a single cavity with aspect ratio W/H equal to 2 (W=0.212m and H=0.106m) (Figure 1a). Vertical profiles of the mean wind field (u and w components), and the turbulent kinetic energy, k, measured upstream of the cavity were specified as the inlet conditions 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.
The experiment investigating the airflow around a wall-mounted cube 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 in full scale (Figure 1b). The vertical profile of the mean wind field measured upstream of the obstacle at $X/H=-8$ was used to derive the vertical profile of u at inflow, while for the turbulent kinetic energy, k, and its dissipation rate, ε, the approximations for a developed boundary layer were used at inflow. Vertical profile of the u-component at different positions (X/H=–1.5, –0.625, 0, 0.625, 1.5 and 2.5) in the centre plane of the flow were available.
A very comprehensive field database was obtained in Göttinger Strasse in Hanover. This is a four-lane street canyon 25 m wide, with buildings ca. 20 m high leading to an aspect ratio W/H=1.25. The obtained database included street and background concentrations and meteorological data from a 10-meter mast on top of a nearby building as well as traffic counts (Figure 1c). In addition wind tunnel measurements performed at the University of Hamburg

(Liedke et al. 1998 and Chauvet et al. 2000) are available for both the concentration at the receptor point and the flow field inside the street canyon.

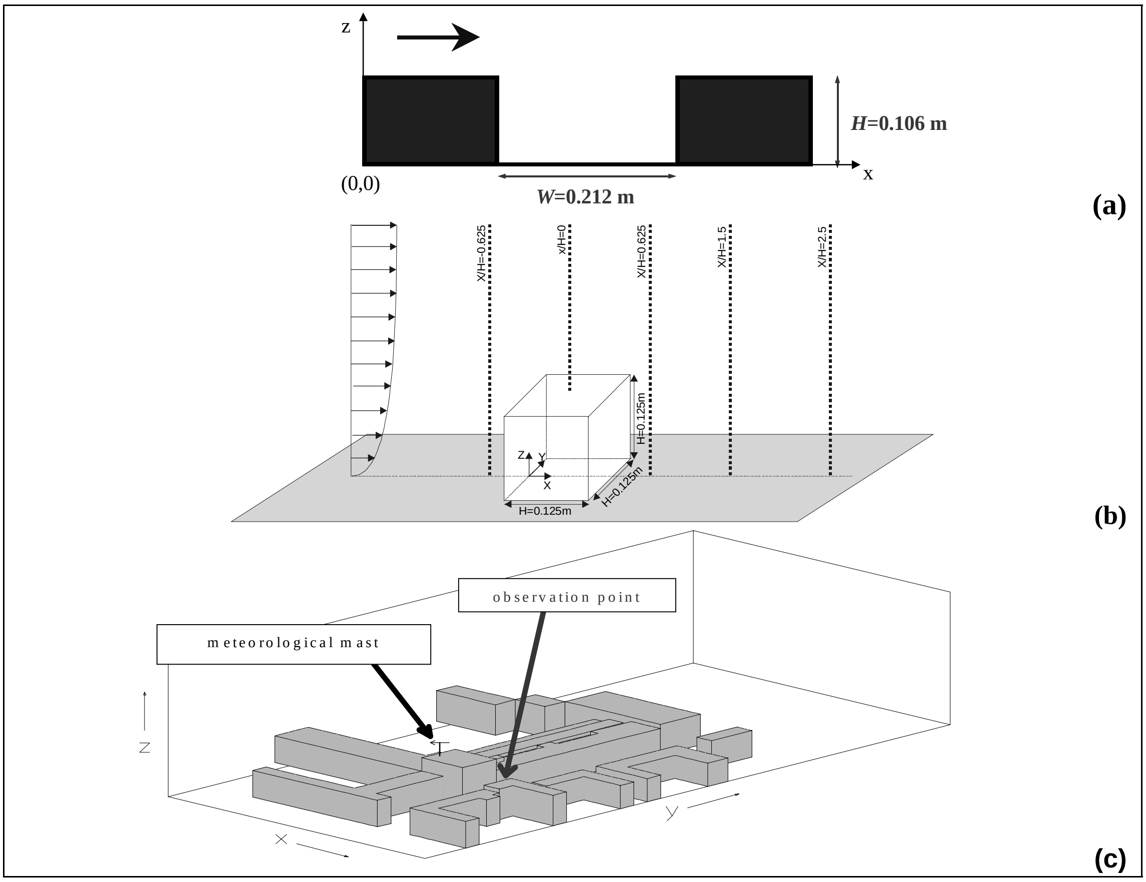

Figure 1: Schematic representation of **(a)** the single cavity, **(b)** the surface mounted cube, and **(c)** the real street.

3. RESULTS

3.1. Single cavity

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. Figure 2a shows the general flow pattern observed within the cavity and 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. 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 varying between –0.15m/s and 0.15m/s.

Figure 2b shows the vertical profile of the *u* component at *X/W*=0.1, 0.5, and 0.9. The numerical results and the measurements for the mean wind field at the different positions within the cavity are overall in a fairly good agreement. Nevertheless, a focussed examination of the profiles close to the solid boundaries (building walls and ground) shows that the models differ in their predictions. The detailed examination of the source code showed that the origin of this difference is mainly the different implementation of the wall function. It is generally observed that two groups of two models, namely, CHENSI and CFX-TASCflow as well as CHENSI-2 and MIMO following the same wall-function implementation predict very similar velocity values close to the walls and ground. 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 main feature of the intercomparison is that all models predict similar velocity values at locations of weak local flow, e.g. at *X/W*=0.1, and the comparison with the data is fairly good. On the contrary, at *X/W*=0.5 and 0.9 where the flow is stronger the discrepancies among the model results are larger. CHENSI and CFX-TASCflow underestimate the velocity close to solid boundaries, while CHENSI-2 and MIMO show better agreement with the data at all examined positions. Therefore, it is suggested that the effect of the different wall-function implementation by the codes on the calculated velocities is small at locations of weaker airflow, while, as CHENSI-2 and MIMO showed better agreement with the data, the implementation of the wall-function applied by these two models is probably more appropriate for similar studies in cavities. Well above the cavity model results and data are exactly matched as the effect of the cavity on the airflow is small and the flow is in equilibrium with the surface below. The same conclusion was reached when simulating a simple boundary layer in equilibrium with a rough surface (not shown here).

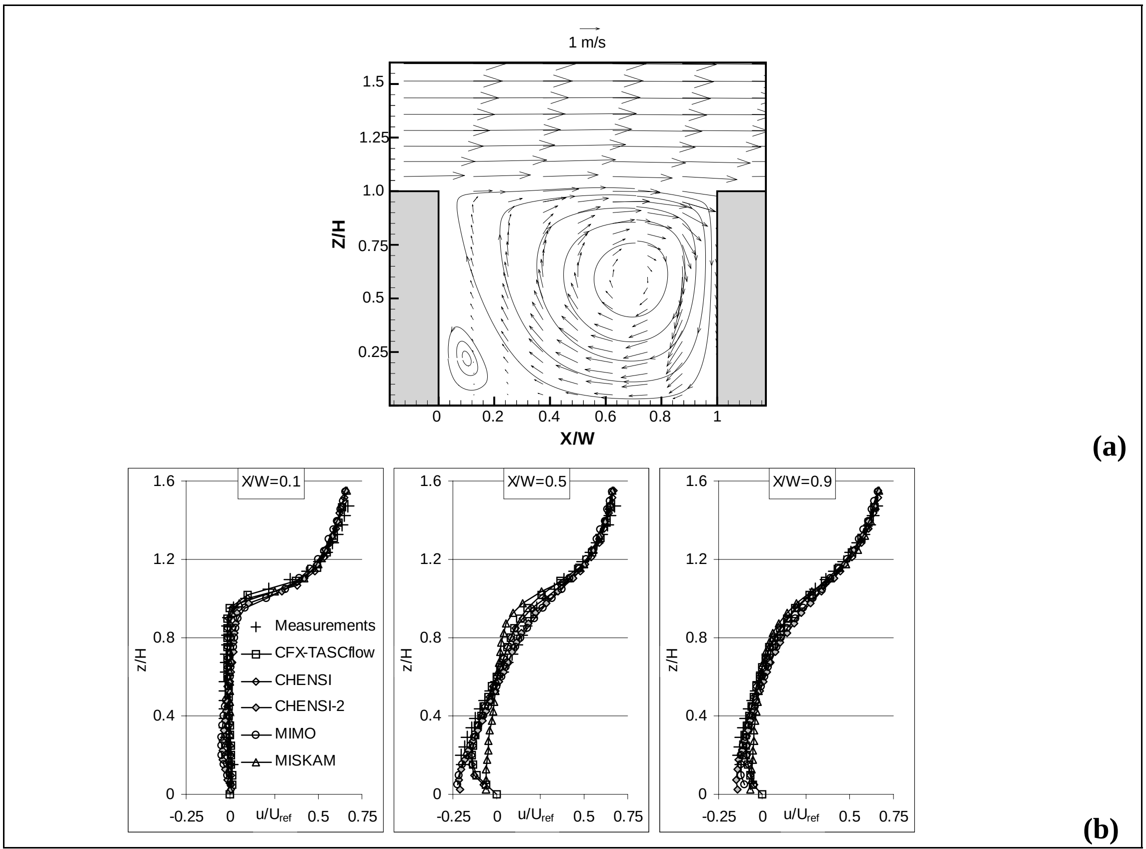

Figure 2: (a) The flow field within and above the cavity as it was reproduced by CHENSI, **(b)** Comparison of the profile of the u-component observed in the wind tunnel with that predicted by the numerical models at *X/W*=0.1, 0.5, and 0.9.

3.2. Surface mounted cube

The wall-mounted cube case was defined as the geometrically simplest three-dimensional case to investigate the performance of the codes in reproducing the flow field around an idealised building. Figure 3a shows the vertical cross section of the dimensionless *u*-component normalised with the free-stream velocity (U_{ref} = 6 m/s) as predicted by MIMO in the centre plane of the flow. In agreement with the findings of the experiment, all models fairly reproduce the airflow pattern. The oncoming flow exhibits an impingement region at the windward side of the obstacle and the increasing pressure leads to the development of a main horseshoe vortex wrapping around the cube. At the upper leeward edge of the obstacle the flow separates leading to a wake region behind the cube that interacts with the horseshoe vortex.

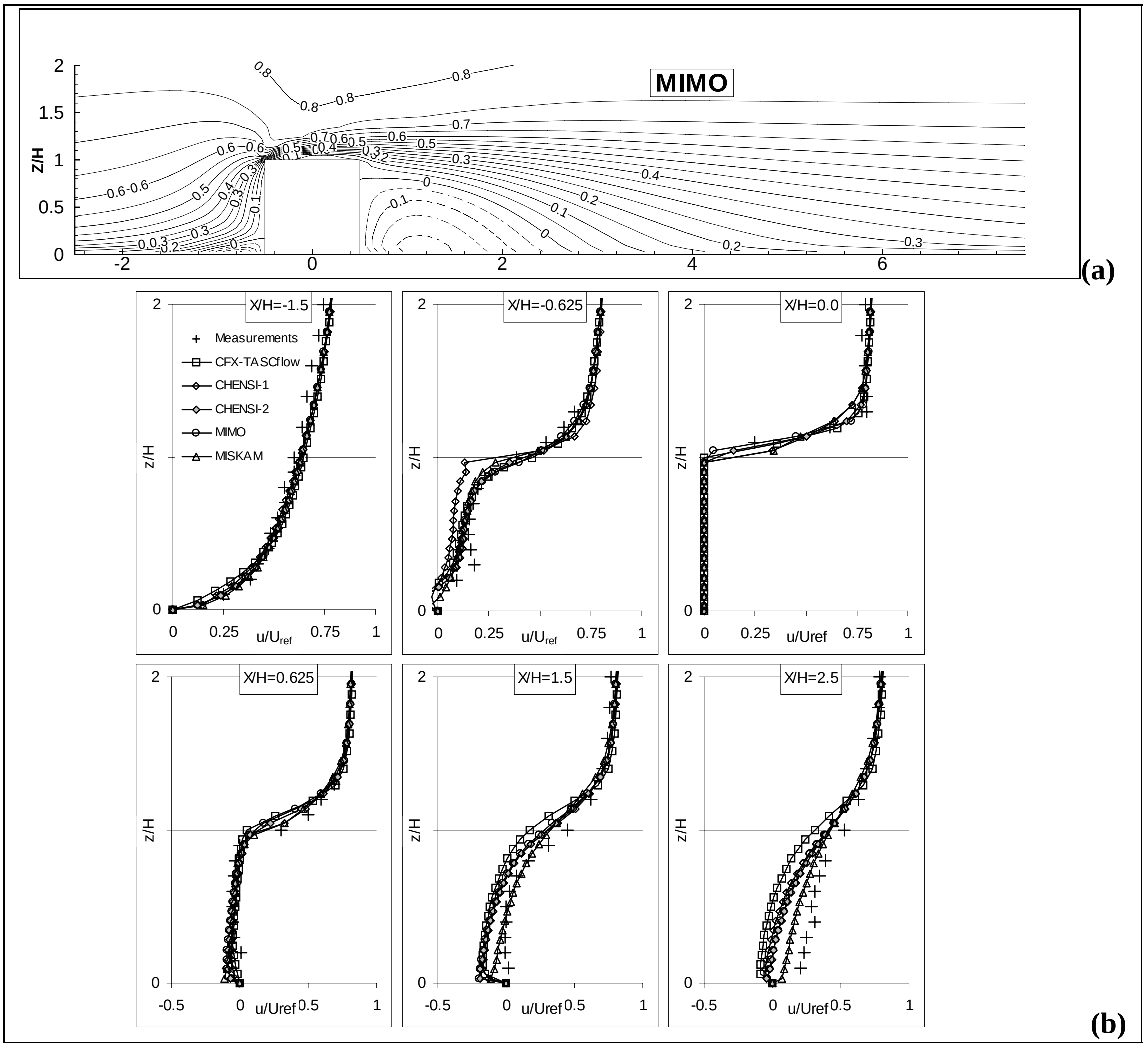

Figure 3: (a) Vertical cross section of the dimensionless u-component predicted by MIMO in the centre plane of the flow, **(b)** Comparison of the dimensionless profile of the u-component measured and predicted by the models in the centre plane of the flow at various distances.

Figure 3b demonstrates the level of agreement among the model predictions and the measured values for the vertical profile of the *u* component 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 and thence the models have managed to simulate the exact experimental conditions prevailing upwind of the obstacle. The comparison between the numerical results and the measurements at the other *X/H* positions 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 wake 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. This overprediction of the re-circulation in the wake is a feature that standard *k-ε* models present and, from the air quality point of view, could lead to overestimation of the transport of pollutants at distances farther downstream than in reality. All models perform similar upstream the obstacle except from CHENSI, which has been generally found to underestimate the velocity at strong local flow regions (Kovar-Panskus et al., 2001). The main reason for this underestimation is the method of implementing the wall functions in this model. 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 as it was also found in the single cavity case.

3.3. Real street canyon

Figure 4 shows the airflow in the horizontal plane at *Z*=10m as predicted by the five models within Göttinger Strasse for wind direction 260° from the North, which corresponds to approaching wind perpendicular to the street. Although the aspect ratio of the street, *W*/*H*=1.25, would imply that a mean re-circulation should develop within the street for perpendicular wind (Oke, 1987), neither any of the models nor the wind tunnel results showed such a simple flow pattern. Both predicted and measured flows show complicated features mainly characterised by vortices developed at the corners of the buildings as well as air entrainment from the side streets, while the main flow pattern shows a strong flow parallel to the street in its northern part. These features are predicted by all models and generally agree with the wind tunnel observations.

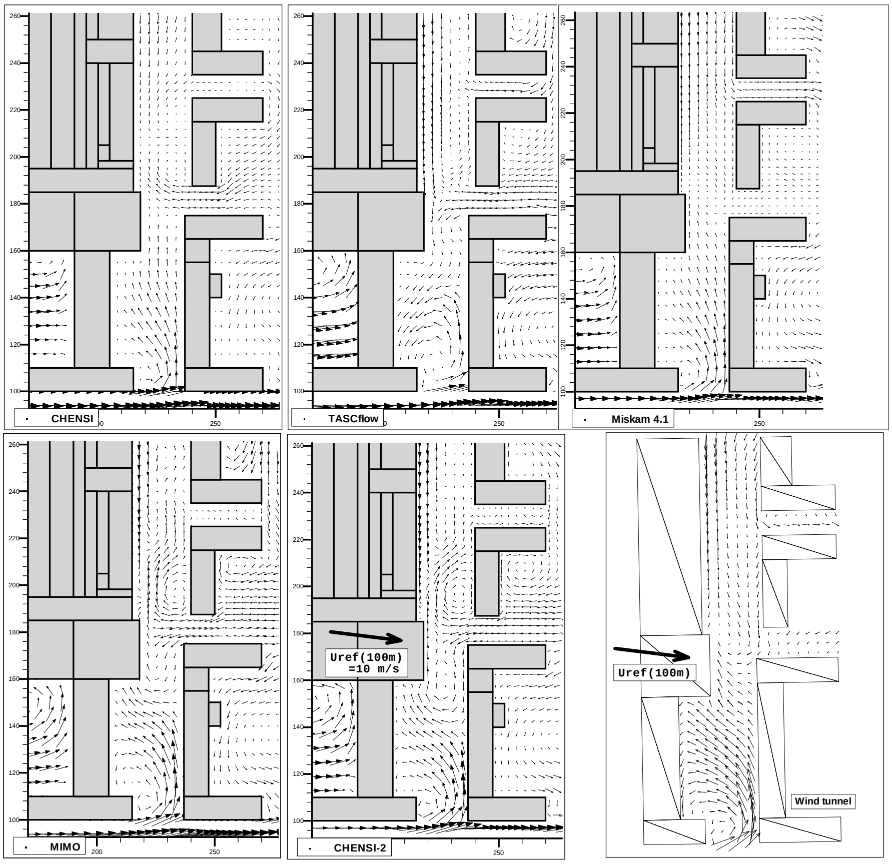

Figure 4: Flow field in horizontal plane calculated by the CFD models and measured in the wind tunnel (wind direction 260° from the North).

Focusing on the flow pattern in the vicinity of the location of the concentration measurements, the models show some differences mainly in the location of the centre of the vortex produced in this area as well as in the intensity of the flow. These, generally small, differences have a large impact on the calculated concentrations. Figure 5 shows the normalised concentration $C^* = (C\,V_{ref}\,H_{ref})/(Q/L)$, where C is the actual concentration, V_{ref} is a scaling velocity (10m/s taken at a reference height of 100m), H_{ref} is a scaling height (20m, the average height of the buildings), Q is the source strength and L its length, versus the wind direction. Agreement is found for the general shape of the graph for all codes. However, for specific wind directions the calculated concentration may vary by a factor 2 to 7. For wind direction 280°, the predicted C^* is in a range of values between 18 and 110, i.e. there is a difference of a factor of 6, even though the general characteristics of the simulated flow were similar. This example illustrates the large difficulty in predicting the pollution concentration at a location that lies in a region with complex flow patterns and strong gradients. Thus any small difference in the predicted direction of the flow impinging at the measuring location may lead to a large discrepancy among the values predicted by similar codes.
These results suggest that the accuracy of CFD modelling for only one location affected by local gradients should be treated with special care. It is also recommended to experimentalists to avoid monitoring the air quality of streets close to irregularities in the building configuration (intersections, gateways, towers, corners etc.). In addition, as the grid resolution has been always an important issue in CFD modelling, it is expected that it may also explain the discrepancies between model results and data obtained in complex configurations.

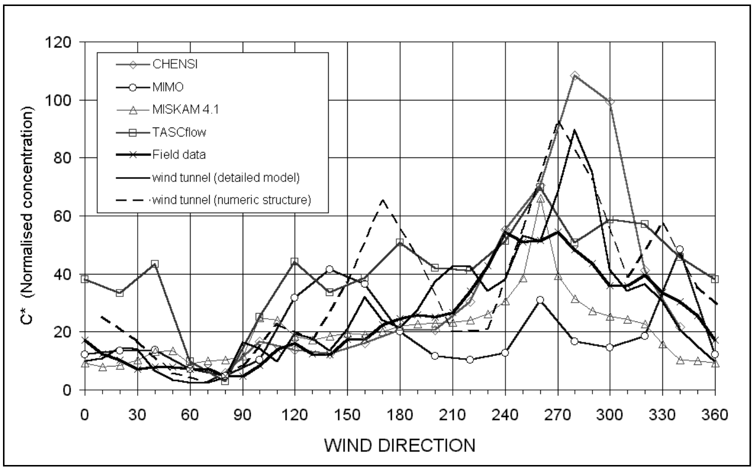

Figure 5: Normalised concentrations versus the wind direction calculated by 4 CFD codes and measured in the full-scale experimental site and in the wind tunnel.

4. CONCLUDING REMARKS

Improvement and optimisation of the methods used in practical application of traffic pollution models for air quality assessment studies are the final goals of the TRAPOS network. The present CFD intercomparison exercise provided the necessary input for achieving this goal. The results of the model intercomparison indicate that CFD models properly describe the overall flow pattern in the urban atmosphere with simple and complex building arrangements. However, a detailed examination of the model predictions close to the solid boundaries suggested that the implementation of the wall-function influences the calculated velocity values especially at locations where the local flow is strong as was found for the simpler geometrical configurations investigated (single cavity and cube). The effect of the wall-function implementation was also obvious in the more complex case of Göttinger Strasse partly leading to differences in the location of the centre of vortex structures and the intensity of the flow. Consequently the calculated concentration of vehicular exhausts at a location

close to building irregularities showed large differences that may reach a factor of 7. In addition, the grid resolution may have its own impact on the discrepancies between model results and data obtained in complex configurations.
It is suggested that CFD codes should not be used for a very local purpose (e.g. a single observation point). The accuracy of the CFD modelling results for only one location affected by local gradients should be treated with special care. For practical purpose, it is likely that an estimation of averages in time (over different inflow situations) or averages in space (to avoid local gradients) is more appropriate. In addition, CFD modelling may be used for recommendations for detecting suitable monitoring sites in order to obtain a representative picture of the air quality in a street canyon. The strong local gradients observed close to irregularities in the building configuration (intersections, gateways, towers, corners etc.) should be avoided, while a variety of measuring positions should be considered.

REFERENCES

1. Chauvet, C., Leitl, B., Schatzmann, M. (2000), 'Wind tunnel modelling of microscale dispersion in a street canyon', *16th IMACS World Congress 2000,* August 21-25 2000 Lausanne, Switzerland.
2. Ehrhard J., Khatib I.A., Winkler C., Kunz R. Moussiopoulos N., Ernst G. (2000), 'The microscale model MIMO: development and assessment', *J Wind Eng. Ind. Aerodyn.* **85**, 163-176.
3. Eichhorn, J. (1989), 'Entwicklung und Anwendung eines dreidimensionalen mikroskaligen Stadtklima-Modells', Dissertation, Universität Mainz, Germany.
4. 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.
5. Kovar-Panskus, A., Louka, P., Mestayer, P.G., Savory, E., Sini, J.-F. and Toy, N.: 2001, Influence of geometry on the flow and turbulence characteristics within urban street canyons – Intercomparison of wind tunnel experiments and numerical simulations, *Proceedings of the 3rd International Conference on Urban Air Quality*, 19-23 March, 2001, Loutraki, Greece.
6. Liedke, J., Leitl, B., Schatzmann, M. (1998), 'Car exhaust dispersion in a street canyon - Wind tunnel data for validating numerical dispersion models', *2nd East European Conference on Wind Engineering (EECWE)*, Prague, 7-11 September 1998.
7. Oke., T.R. (1987), Boundary Layer Climates – 2nd edition, Methuen, UK.
8. Raw, M.J., Galpin, P.F., 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.
9. Sini, J.-F., Anquetin S., Mestayer P.G. (1996), 'Pollutant dispersion and thermal effects in urban street canyons', *Atmos. Env.* **30**, 2659-2677.
10. URL1: TRAPOS URL http://www.dmu.dk/AtmosphericEnvironment/trapos/
11. URL2: CFD working group URL http://www.dmu.dk/AtmosphericEnvironment/trapos/cfd-wg.htm

Acknowledgements

The model intercomparison presented was carried out in the frame of the research network TRAPOS operating in the frame of European Commission Training and Mobility of Researchers Programme. Also, the provision of input data by Drs Savory and Kovar-Panskus of the University of Surrey and Drs Schatzmann, Liedke and Chauvet of the University of Hamburg and Dr. Müller of the State Environmental Agency of Lower Saxony (NLÖ Hanover) is acknowledged. 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.