ELSEVIER

Journal of Wind Engineering
and Industrial Aerodynamics 67&68 (1997) 293–304

JOURNAL OF wind engineering AND industrial aerodynamics

Car exhaust dispersion in a street canyon. Numerical critique of a wind tunnel experiment

Bernd M. Leitl[1], Robert N. Meroney

Fluid Dynamics and Diffusion Laboratory, Civil Engineering Department, Colorado State University, B-227 Engineering Research Center, Fort Collins, CO 80523, USA

Abstract

Due to increasing car traffic in cities, problems related to car induced air pollution in street canyons have become important. Physical modeling in wind tunnels or numerical codes may be used for dispersion simulation when investigating air quality. Rafailidis [in: Annalen der Meteorologie] carried out an extensive set of test runs recently in the BLASIUS wind tunnel at the Meteorological Institute of the University of Hamburg, Germany. In the present study the wind tunnel experiments were simulated numerically using the CFD-code Fluent®. In a first approach, the idealized two-dimensional case was calculated. Several test runs have been carried out to study the effect of emission rate and source design on flow structures and dispersion in the street canyon. It could be shown that alternative emission conditions and the source design might affect the concentration field within a modeled street canyon. A second set of calculations for a simplified three-dimensional simulation of the street canyon setup was performed to investigate the presence of secondary flow patterns found during wind tunnel tests. The lateral flow structure within the street canyon observed during wind tunnel measurements was simulated, and the effect of changing boundary conditions on the secondary flow structure was studied. In the paper the advantages of CFD simulations for planning wind tunnel dispersion tests are discussed.

Keywords: Dispersion; Wind tunnel; Street canyon; Numerical simulation

1. Introduction

Increasing car traffic leads to increasing air quality problems in cities. Since cars are accepted to be the major emission source of air pollutants in urban areas, and

[1] Present address: Meteorology Institute, University of Hamburg, Bundesstrasse 55, D-20148 Hamburg, Germany.

PII S0167-6105(97)00080-9

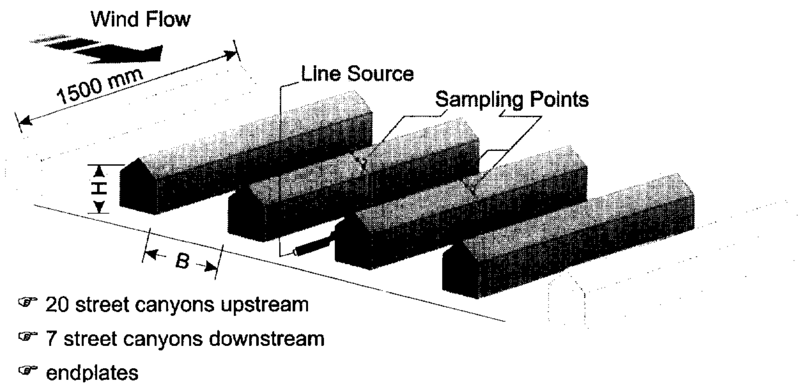

Fig. 1. Wind tunnel setup.

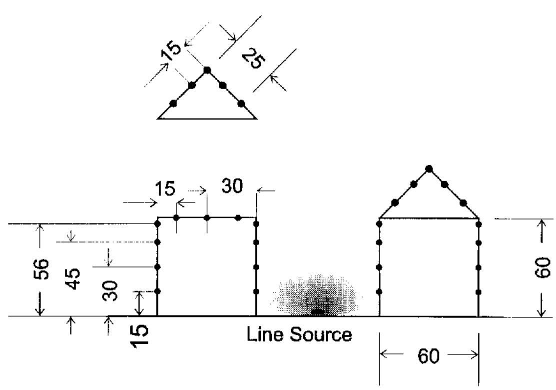

Fig. 2. Configuration of measurement tapping holes at the outline of the test street canyon.

a further increase of city traffic is expected, investigations of dispersion processes in street canyons have become a focal point in environmental research (see Refs. [1, 2]). Recently Rafailidis et al. [3] carried out an extensive set of wind tunnel experiments on gas dispersion in urban street canyons. Using a quasi two-dimensional setup (see Fig. 1) the dispersion of tracer gas emitted by a line source in a closed street canyon was measured for a variety of canyon aspect ratios, B/H, and roof forms. To simulate an urban roughness 20 upstream and 7 downstream street canyons were included. The concentrations were measured in the symmetry plane of the setup at the positions shown in Fig. 2. A more detailed description of the wind tunnel experiments and the physical model is given in Refs. [3, 4]. A significant lateral flow structure (Fig. 3) was detected and measured in the street canyon, where the flow was expected to be quasi two-dimensional. Since the secondary flow structure obviously causes a concentration

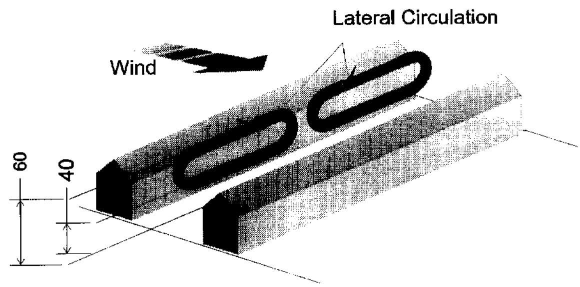

Fig. 3. Lateral flow structure observed during wind tunnel experiments.

gradient in the lateral direction, it was also expected to affect the concentration measurements in the symmetry plane. Extensive flow visualization experiments using a Laser light sheet visualization setup and Laser-Doppler-Velocimeter measurements were performed in order to locate the source, but a final statement about what was causing the 3D flow structure and how to minimize the influence of lateral flow within the street canyon could not be made. Also, it was difficult to predict the magnitude of the influence of the second order flow on mean concentrations measured in the center plane of the street canyon.

A numerical simulation with the CFD-code Fluent® was prepared to give additional information about the mean flow within the street canyon as well as the concentration distribution for a quasi two-dimensional street canyon setup. The Fluent CFD package consists of several tools for defining a discrete flow problem (i.e. grid generation), setting boundary conditions and solving the set of complex equations for conservations of momentum, mass, chemical species and energy. The governing equations are discretized on a curvilinear grid to permit computations over irregular geometries and solved using a control volume based finite difference method. Discrete velocities and pressures are stored in a nonstaggered grid, and interpolation is realized by using a first-order, power-law scheme or optionally by higher-order upwind schemes. The basic solver in Fluent is the SIMPLEC/SIMPLE algorithm with iterative line-by-line matrix solver and multigrid acceleration. For a detailed description of Fluent see Ref. [5].

2. Calculations

2.1. Two-dimensional calculations – setup

In a first approach the flow in the street canyon was assumed to be two dimensional. Several discrete 2D setups with different body-fitted grid configurations were tested with respect to their numerical stability before a final grid was selected. An optimum of performance according to memory usage, convergence behavior and computing time could be found for the grid in Fig. 4 which consists of 230×112 cells

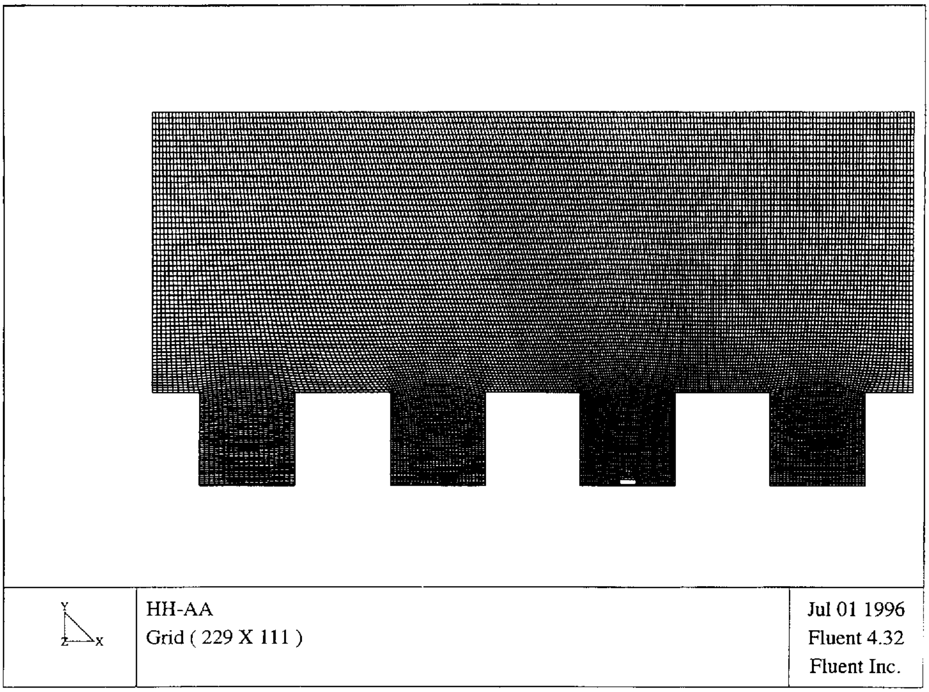

Fig. 4. Computational grid – case A, flat roof configuration.

in a physical domain of 480 mm × 240 mm. The inlet boundary (left side) was defined at the middle of the 3rd up stream building where the wind profile was measured during wind tunnel experiments; hence, the wind tunnel profiles of velocity and turbulence intensity could be used for calculating boundary conditions for the inlet. The wind tunnel data as well as the input profiles of velocity, turbulent kinetic energy and dissipation used for the numerical simulation are shown in Fig. 5. The inlet values of kinetic energy, k, and the dissipation ratio, ε, were calculated from measured velocity profiles and turbulence intensities as well as from a given friction velocity ratio, u_*/u_{ref}, for the wind tunnel setup according to Eqs. (1) and (2) (y is the distance from the wall, κ is the von Karmán constant),

$$\varepsilon = u_*^3/\kappa y, \tag{1}$$

$$k = \tfrac{3}{2}(u')^2. \tag{2}$$

The line source inlet was modeled as a dx = 0.2 mm wide slot (0.25 mm tubing in the wind tunnel) and set as a constant velocity inlet, no turbulence and an inlet velocity of $w_{\mathrm{source}} = 3.81$ m/s according to the source emission rate used during the wind tunnel experiments. When turbulent mixing was simulated, a tracer mass fraction of 1 was applied to the line source inlet. The wind tunnel situation, where 1 l/h ethane was mixed in 100 l/h air was also tested. As expected, the nondimensional

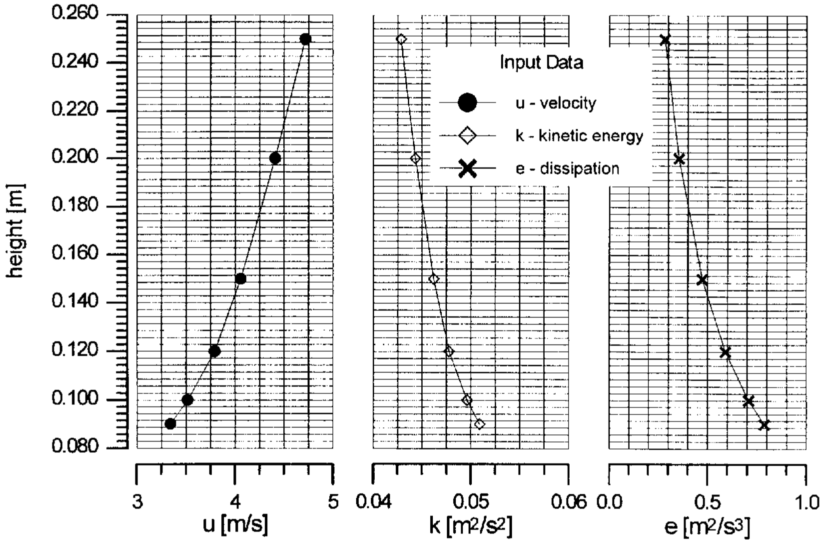

Fig. 5. Inlet profiles of velocity, kinetic energy and dissipation.

concentration values show no dependency on which source concentration is used. The top and right side of the computational domain were defined as outlets to allow for expected blockage effects caused by the models. The model buildings as well as the wind tunnel floor were specified as hydraulic smooth, isothermal walls with $u = w = 0.0$ m/s (u is the horizontal velocity component, w is the vertical velocity component).

To study the influence of applying different turbulence models in Fluent, all configurations were calculated with a standard k–ε model as well as with the newer RNG model (ReNormalized Group theory, see Refs. [5, 6] for description). As recommended by Van Oort and Stork [7] the constants of the standard k–ε model were varied to visualize possible improvements of the numerical results compared to the wind tunnel experiments for different constant sets. According to recommendations given in Ref. [5] the turbulence model constants were not changed when the RNG model was used.

Since the wind tunnel results of concentration measurements were provided in a non-dimensional form all calculated concentrations, C were normalized in the same way with source emission rate ($w_{\text{source}}\mathrm{d}x_{\text{source}}$, where w_{source} is the exhaust velocity, $\mathrm{d}x_{\text{source}}$ is the width of source), reference velocity, U_{ref}, and the total height of the model building, H (see Eq. (3)),

$$K = CHU_{\text{ref}}/w_{\text{source}}\mathrm{d}x_{\text{source}}. \qquad (3)$$

2.2. Two-dimensional calculations – results

Comparisons of the mean velocity fields calculated by a standard k–ε approach versus the velocity field calculated by the RNG approach displayed no major

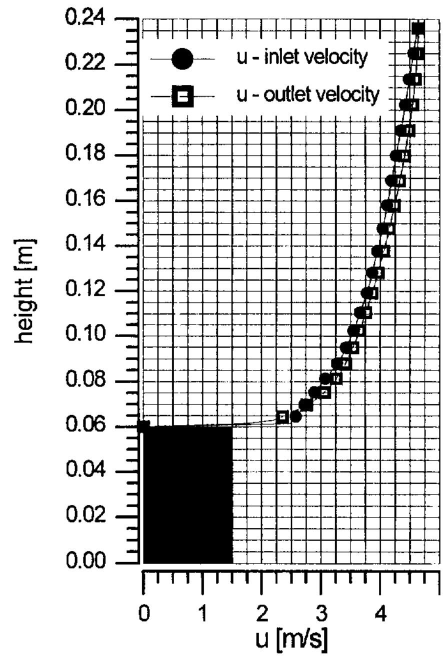

Fig. 6. Comparison between given inlet velocity profile and calculated outlet profile.

differences. In Fig. 6 the given inlet velocity profile (i.e. measured velocity profile) for the flat roof configuration (case A) is compared to the calculated outlet profile. Both profiles are almost identical except for a slight smoothing effect close to the building roofs and the effect of the additional mass source flow. Even if the exhaust velocity of tracer is high compared to the mean velocity of the surrounding canyon flow, the source injects the tracer into the street canyon without major disturbance when operating the source at 101 liter tracer per hour.

Fluent computes similar concentration patterns observed during wind tunnel and field measurements. The main vortex in the canyon dominates the dispersion of car exhaust and generates higher concentration values on the upwind or downwind building for different configurations. Depending on the roof geometry, the flow pattern and the location of the main vortex in the street canyon changes. This causes a change in measured/calculated concentrations at the side walls of the street as well. Fig. 7 shows a direct comparison between measured and calculated concentrations over the upwind and downwind walls for the flat roof configuration (case A). The concentrations calculated for the upwind wall of the street canyon agree well with those measured in the wind tunnel. No major difference can be detected when comparing calculations based on the standard k–ε-model with results from a RNG-simulation except for an area close to the source. At the downwind wall the numerical simulation leads to significantly lower values than measured in the wind tunnel.

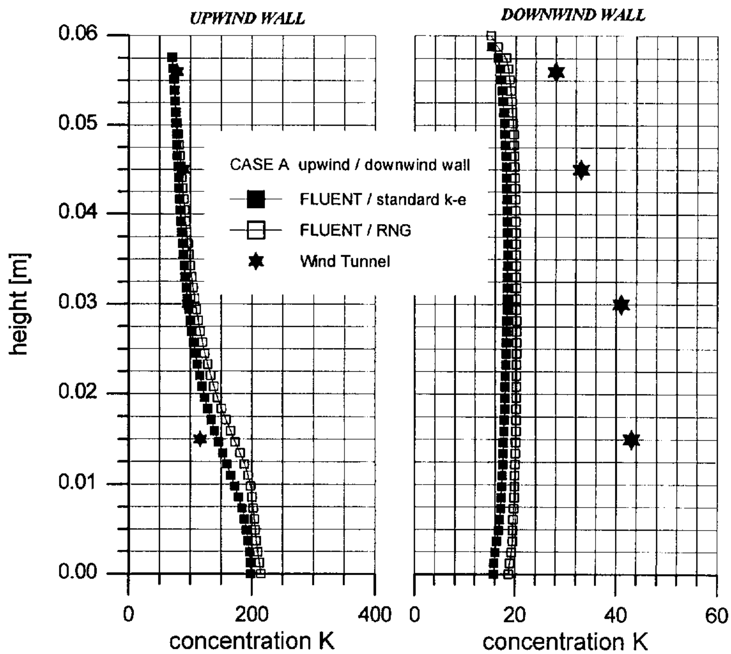

Fig. 7. Comparison between measured and calculated concentrations.

Again the k–ε-model and the RNG-model give approximately the same tendency, but an 'offset' concentration of about 10% for the RNG-model can be detected. Compared with the wind tunnel results the numerical simulations show a difference of up to 58% for the lowest measurement point within the street canyon. For slanted roof configurations (case B: one slanted roof upstream the street canyon, case C: slanted roof on upwind and downwind building) even bigger differences between wind tunnel experiments and numerical simulation were observed. The maximum relative difference based on wind tunnel results was found for case B with values up to 89%. A canyon pollution factor, K^* (K averaged over all sampling points in the canyon), was used to compare the concentration fields for different configurations with the overall K^* values from wind tunnel experiments. As shown in Fig. 8 the averaged concentration value is almost the same in wind tunnel experiment and numerical simulation for the flat roof urban roughness. For slanted roofs the K^* varies up to 50%.

Better agreement between wind tunnel data and numerical simulation could be achieved for slanted roof configuration by changing the constants of the k–ε turbulence model even though there was no general tendency for better results using a modified constant set for all three test cases. Since the calculation of the dispersion within the street canyon is dependent on location and size of the recirculation zones even slightly different calculations of the velocity field had a dramatic effect on the concentrations calculated at the reference points. The results have shown that the

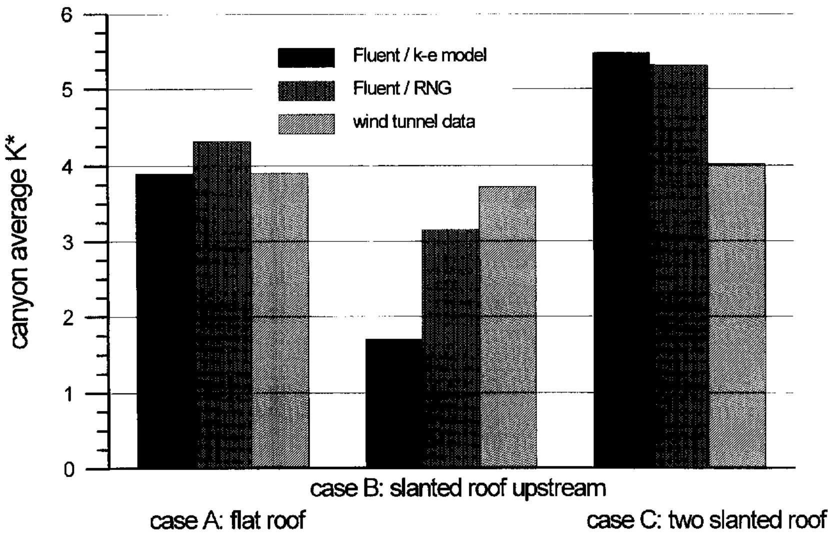

Fig. 8. Canyon pollution factors.

standard constant set cannot be assumed to be universal for complex, high turbulent shear flows with large recirculation areas. Further investigations are required to evaluate constant sets that might lead to a better reproduction of turbulent boundary layer flows and gas dispersion in recirculating areas. However, another reason for discrepancies between the two sets of results might be the lateral flow in the street canyon during wind tunnel experiments studied in the second part of this paper.

2.3. *Three-dimensional calculations – setup*

To study the three-dimensional flow effects in the street canyon observed in the wind tunnel a simplified 3D setup was generated. A body-fitted mesh with $30 \times 33 \times 61$ (60390) cells were used to define the discrete problem for slanted roof configurations with canyon width ratios $B/H = 1$ and $B/H = 0.5$. To avoid grid skewness problems only a few iterations for grid smoothing were applied. The outline of the geometry as well as two meshed surfaces are shown in Fig. 9. To keep the geometrical resolution of the grid high for a limited total number of cells no upstream or downstream street canyon was included into the discrete model. The flow pattern within the street canyon is dominated by the geometry of the street canyon itself and the separation at sharp edges on the roof. The mean flow field calculation could be assumed to be independent from upstream/downstream street canyons. The inlet profiles for velocity, kinetic energy and dissipation were taken from a 2D simulation with several upstream canyons modeled upwind of the source canyon, and the inlet conditions were derived from wind tunnel measurements (see Section 2.1).

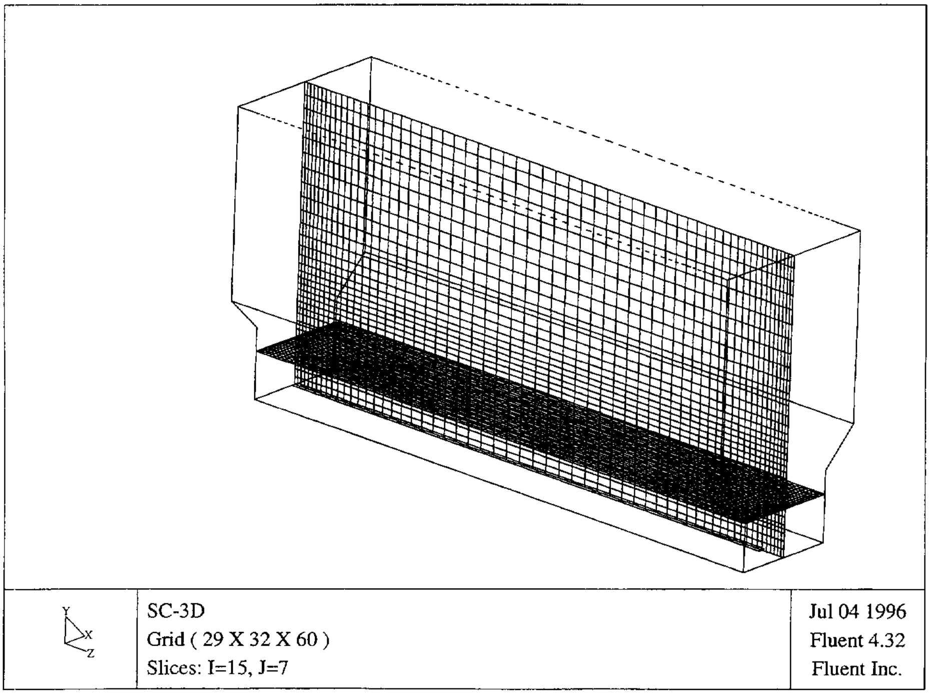

Fig. 9. Simplified 3D street canyon setup.

2.4. Three-dimensional calculations – results

A lateral flow structure could be found during the numerical simulation. It could be shown that the major force for the secondary flow is the corner vortex at the end plates of the model canyon. This vortex structure is present for all configurations with end walls or end plates and is caused by the boundary layer at the side walls as well as by the resulting lateral pressure gradient. Similar flow patterns can be observed in a variety of technical flows in rectangular air-conditioning ducts. For a more narrow street canyon ($B/H = 1/2$) the lateral velocity appears to be slightly higher than for a wider canyon ($B/H = 1$). The variation might be caused by the smaller overall velocity in the narrower canyon which is more easily affected by the pressure field as well as the corner vortex at the end plates. A comparison of measured and calculated velocity profiles for the $B/H = 1/2$ setup is given in Fig. 10. The calculated results agree well with Laser Doppler Velocimeter (LDV) measurements in the wind tunnel. Except in an area close to the side walls the resulting concentration field shows no major deviation. The effect of the lateral circulation seems to be small in terms of higher or lower concentrations in the symmetry plane, where the sampling points were located during the wind tunnel experiments. When comparing the calculated concentrations with the results of wind tunnel experiments, the differences are in the same

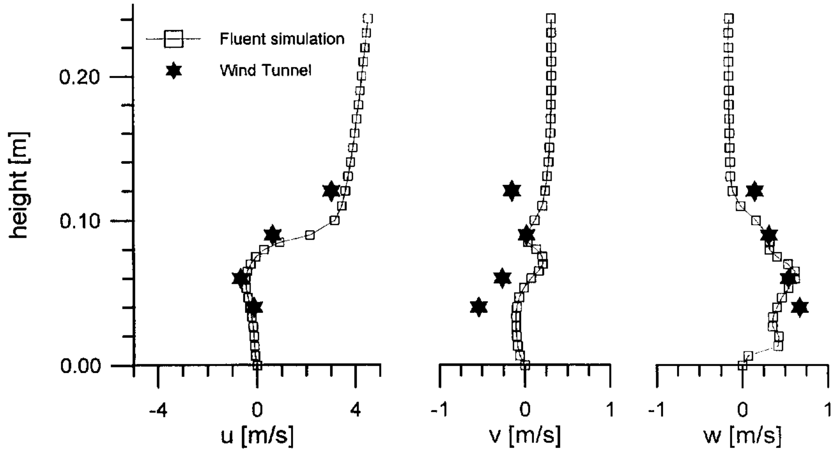

Fig. 10. Comparison between measured and calculated velocity components – 3D simulation.

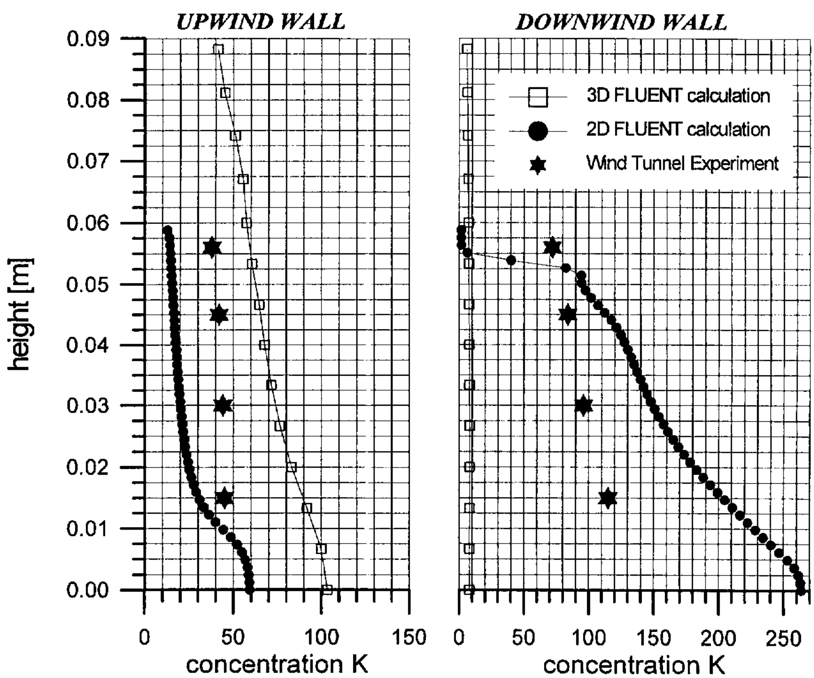

Fig. 11. Calculated and measured concentration profiles – 3D simulation.

order of magnitude as for 2D simulations at the upwind wall (max. 90%). A totally different concentration profile was calculated for the downwind wall. In a 3D simulation an almost constant concentration at a very low level is predicted for the downwind wall where the typical canyon profile (i.e. higher concentrations at the lower elevations) was found during 2D simulations as well as in the wind tunnel (see Fig. 11, case $B/H = 1$).

3. Conclusions

It has been shown that the FLUENT code can simulate the flow field in urban street canyons. Mean flow patterns as well as predicted separation/recirculation areas agree well with the results from physical modeling in a boundary layer wind tunnel. Complex 3D flow structures like secondary vortexes found in the physical model could be simulated, and the calculated velocity components agree reasonably well with LDV measurements from the wind tunnel experiment. The effect of changing source design, source emission rate or wind speed can be simulated, and the resultant changes in the flow field can be predicted.

Standard k–ε model and RNG turbulence modeling give almost the same results for the calculated flow pattern within a street canyon that is bounded by sharp edged buildings. Discrepancies between results from wind tunnel experiments and numerical simulations were found for the predicted concentration fields in a street canyon when simulating car exhaust dispersion. Independent of the turbulence model used the concentration results show differences up to 90%. Using numerical codes like Fluent can help to design and setup wind tunnel experiments; hence reducing the time required to optimize a physical model and expensive pre-runs in a wind tunnel. With a numerical simulation critical points like source design for dispersion simulation can be examined and boundary conditions can be modified.

Acknowledgements

The authors wish to thank Dr. Stilianos Rafailidis (University of Hamburg, Germany) for providing the model drawings and wind tunnel test results for comparisons that made this study possible. Also the support by the Alexander von Humboldt Foundation is gratefully acknowledged.

References

[1] R.N. Meroney, M. Pavageau, S. Rafailidis, M. Schatzmann, Study of line source characteristics for 2-D physical modelling of pollutant dispersion in street canyons, J. Wind Eng. Ind. Aerodyn. 62 (1996) 37–56.

[2] M.C. Murphy, A.E. Davies, Wind tunnel modelling of vehicle emissions on roadways as line sources, in: Proc. Air Pollution Control Association Conf., Dallas, TX, June 1988.

[3] S. Rafailids, M. Pavageau, M. Schatzmann, Wind tunnel simulation of car emission dispersion in urban street canyons, in: Annalen der Meteorologie, Deutsche Meteorologische Gesellschaft, Munich, 1995.

[4] P. Leisen, Windkanaluntersuchungen zur Simulation von Imissionssituationen in verkehrsreichen Straßenschluchten, aus Kolloqiumsbericht Abgasimmissionsbelastungen durch den Kraftfahrzeugverkehr in Ballungsgebieten und im Nahbereich verkehrsreicher Straßen, Verlag TÜV Rheinland, October 1981, pp. 207–234.

[5] Fluent User's Guide, vols. 1–4, Fluent Inc., Centerra Resource Park, Lebanon, NH.
[6] V. Yakhot, S.A. Orzag, Renormalization group analysis of turbulence – I. Basic Theory, J. Sci. Comput. 1 (1) (1986) 1–51.
[7] H. Van Oort, B. Stork, Vergleich von CFD-Berechnungen und Windkanalmessungen der Umströmung einfacher geometrischer Körper, in: E. Plate et al. (Eds.), Windprobleme in dichtbesiedelten Gebieten. D-A-CH '93, Karlsruhe, 1993.