ELSEVIER

Journal of Wind Engineering
and Industrial Aerodynamics 67&68 (1997) 745–755

JOURNAL OF
wind engineering
AND
industrial
aerodynamics

Concentration and flow distributions in the vicinity of U-shaped buildings: Wind-tunnel and computational data

Bernd M. Leitl[a,1], P. Kastner-Klein[b], M. Rau[b], R.N. Meroney[a]

[a] *Fluid Mechanics and Wind Engineering Program, Civil Engineering Department,*
Colorado State University, Fort Collins, CO 80523, USA
[b] *Institut für Hydrologie und Wasserwirtschaft, Universität Karlsruhe,*
Kaiserstrasse 12, 76128 Karlsruhe, Germany

Abstract

The flow and dispersion of gases emitted by point sources located near a U-shaped building were determined by the prognostic model FLUENT using the RNG k–ε turbulent closure approximation. Calculations are compared against wind-tunnel measurements about such a U-shaped building and several other prognostic and diagnostic numerical models. FLUENT gives a mixed image in terms of accuracy of predicted concentrations compared to the wind tunnel experiment. For identical boundary conditions higher as well as lower concentration values are calculated for different test cases. Ground level sources show higher discrepancies than situations where the tracer was emitted from the roof of the model building. A major error source was found to be the stationary solution procedure that was chosen for all simulations.

Keywords: Dispersion; Wind tunnel; Air pollution aerodynamics; Numerical simulation

---

1. Introduction

The flow patterns which develop around individual buildings govern the local distribution of pollution about the building and in its wake. The superposition and interaction of flow patterns associated with adjacent buildings govern the movement of pollutants in urban and industrial complexes. Sources are often located close to buildings, as in short stacks from thermal or chemical processes or ventilation shafts from parking garages.

---

[1] Lynen Research Fellow of the Alexander von Humboldt Foundation, Germany. Present address: Meteorology Institute, University of Hamburg, D-20148 Hamburg, Germany.

PII S0167-6105(97)00115-3

A number of prognostic and diagnostic numerical models have been developed to predict flow and dispersion within the arrangements of buildings, vegetation and vehicle corridors which constitute a city. Prognostic models solve the equations of motion with a selected closure model (mixing length, kinetic energy, Reynolds stress, or sub-grid scale assumptions; e.g. MISKAM [1]; FLUENT 1983 [11]; or LES [2]). Diagnostic models produce mass-consistent 3-D mean wind fields, using intelligent initialization procedures (often based on wind-tunnel model measurements) which analyze the building structure at each grid point and choose typical wind components for the stagnation separation and recirculation zones as a first approximation (e.g. ABC [3]; DASIM [4]; ASMUS [5].) In both cases a dispersion model solves the advection–diffusion equation based on the precalculated wind field and specified exchange coefficients.

Klein et al. [6] compared a number of such models against wind-tunnel measurements around a U-shaped building. The building shape was chosen because it can be represented by a number of alternative combinations of basic rectangular building components. Unfortunately, given such simple rectangular building elements, some diagnostic models produce different flow patterns for the same overall building envelope [7]. Klein et al. [6] found when they compared ABC, ASMUS, DASIM and MISKAM to the wind-tunnel measurements that there were no clear wind, source and/or building configurations where all models succeed or fail. For some wind orientation situations the numerical models produced similar maximum concentrations, but these differed from physical model measurements. In other wind directions even the differences between numerical calculations were huge. Their final conclusion was "it is impossible to recommend one of the models at present". They also noted that it would be necessary to compare local results from the wind field and turbulence parameterization in order to determine the reasons for the differences of the models between one another and the wind tunnel results.

2. Potential of RNG k–ε models

Murakami [2] has compared the calculated flow fields around a cube immersed in a turbulent boundary layer using various turbulence models (standard k–ε, ASM and LES). He concluded that the correspondence between experiment and simulations of the mean wind field "were fairly good, but serious and non-serious discrepancies existed in the pressure and turbulence statistics". He observed that the standard k–ε model overpredicts the value of k in regions of strong shear. This leads to excessive mixing and subsequent incorrect prediction of regions of separation and reattachment.

A renormalized-group-theory (RNG) version of the k–ε model has been proposed which only requires the existence of a large-Reynolds number region of turbulent cascading to generate a set of improved transport equations with no unspecified constants [8]. This model has led to improved predictions of flow and separation around bluff bodies like backward facing steps and cubical bluff bodies. In this paper the flow and dispersion produced by the prognostic model FLUENT using the RNG

k–ε turbulent closure is compared with the measurements from Klein et al. [6] for a U-shaped building.

3. Wind-tunnel measurements about the U-shaped building

The U-shaped building model was presumed to replicate at a scale of 1 : 200 a building with a planform base 40 m by 52 m with a 28 m square courtyard located along the 52 m face of the building. Wind-tunnel measurements of concentration were taken about buildings of heights 16, 28 and 40 m. Gas sources were presumed to lie on the roof of the base of the U, and 20 m up and downwind of this source at ground level (see Fig. 1). Wind approach angles of 0°, 45°, 90°, 120°, 135° and 180° were examined [9].

The experiments were performed in a boundary layer wind tunnel with a test-section about 10.5 m long, 2 m wide and 1 m high. A thick turbulent boundary layer was generated along the floor of the tunnel by a combination of vortex generators and roughness on the floor of the wind tunnel. The meteorological parameters of wind velocity profile, turbulence intensity and turbulence spectra were chosen to be similar in full scale and in the wind tunnel [10]. During the wind-tunnel experiments the power law profile of the approach flow had an exponent of 0.28 and the surface level turbulent intensity reached 33% for a wind speed u_{10} of 5 m/s. Approach wind and turbulence profiles are shown in Fig. 2. Emissions were simulated by releasing a tracer gas (SF_6), and concentrations were detected by analyzing samples with a leak detector. A total of 43 combinations of source location (3), building heights (3), and wind directions (6) were examined.

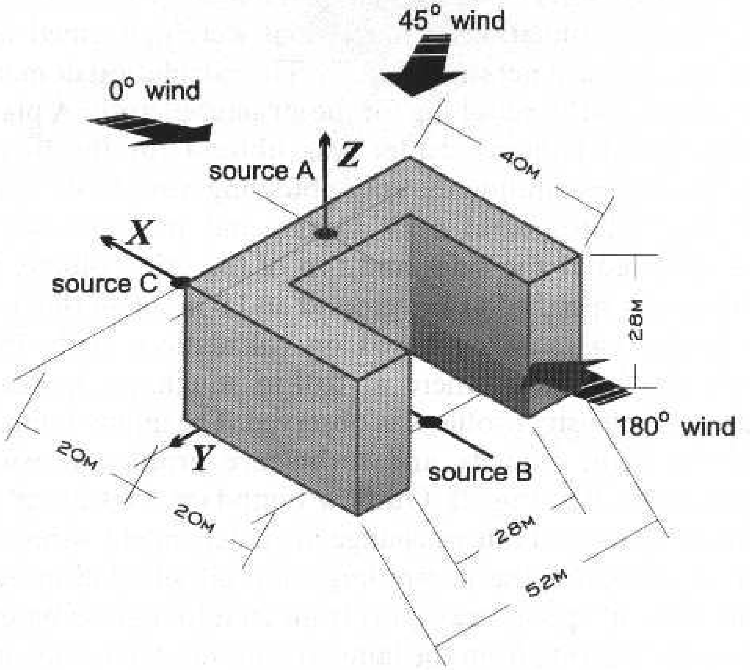

Fig. 1. Configuration of U-shaped building.

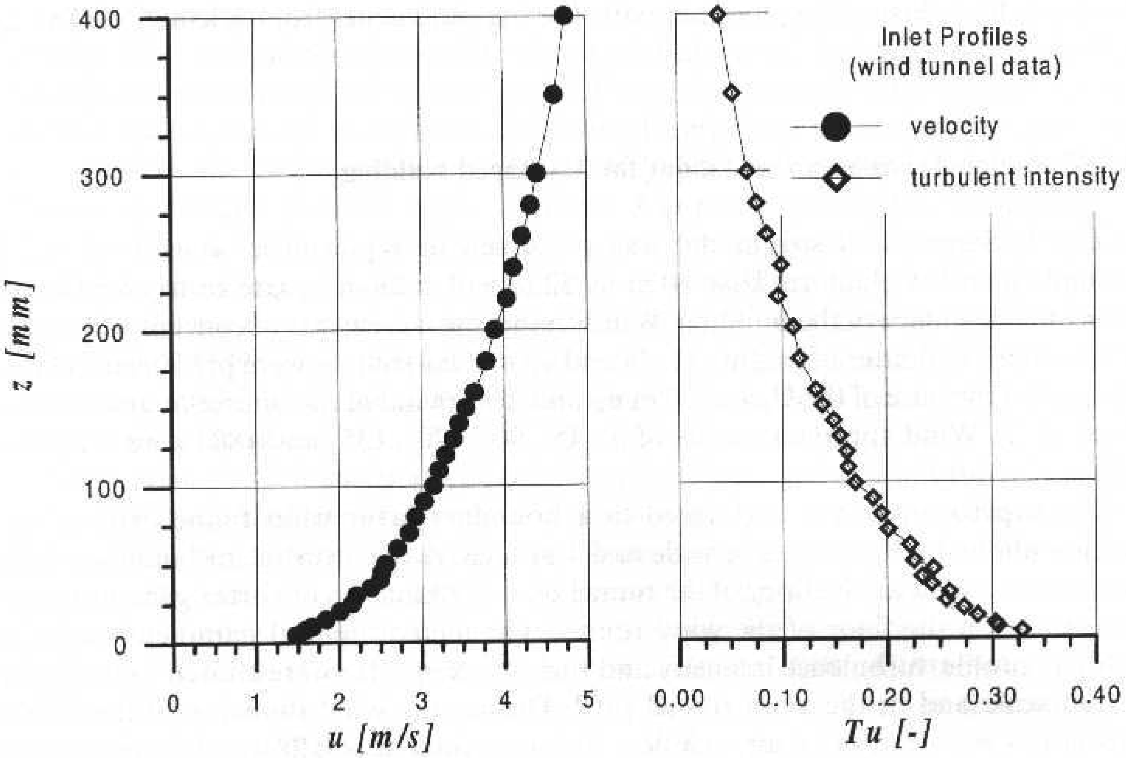

Fig. 2. Mean velocity and turbulent intensity profiles approaching U-shaped building.

4. Numerical model configuration

Version 4.3 of the FLUENT code as well as the new FLUENT/UNS Version 4.0 were used for numerical simulations. Calculations were performed with both structured and unstructured grid generation (Fig. 3). The calculation domain was a grid 94 cells high, 23 cells wide, and 29 cells long for the structured grids. A plane of symmetry through the longitudinal building center was utilized for the 0° and 180° wind orientations. For the 45° orientation the entire building was placed in an unstructured grid volume of 36 875 tetrahedral cells. Outlet and pressure inlet or symmetry boundaries were specified at the sides and top of the grid volume, while a surface roughness of 0.01 m was specified at the ground and 0 m on all the building surfaces. The roughness height was selected based on calculations with different uniform ground roughness (no building) where at 0.01 m roughness height no change in velocity and turbulent intensity profile was observed. The inflow boundary conditions were chosen to match the velocity and turbulence profiles measured during the wind-tunnel experiments (see Fig. 2). Outflow boundary conditions were chosen to maintain constant longitudinal rate of change of all dependent variables (i.e. constant slope). In order to minimize the computing time all simulations were started as laminar solutions without species transport from an initial guess based on the values given at the flow inlet. Starting from the laminar solution the turbulent flow field was calculated in the second step and the species transport was added to the solution

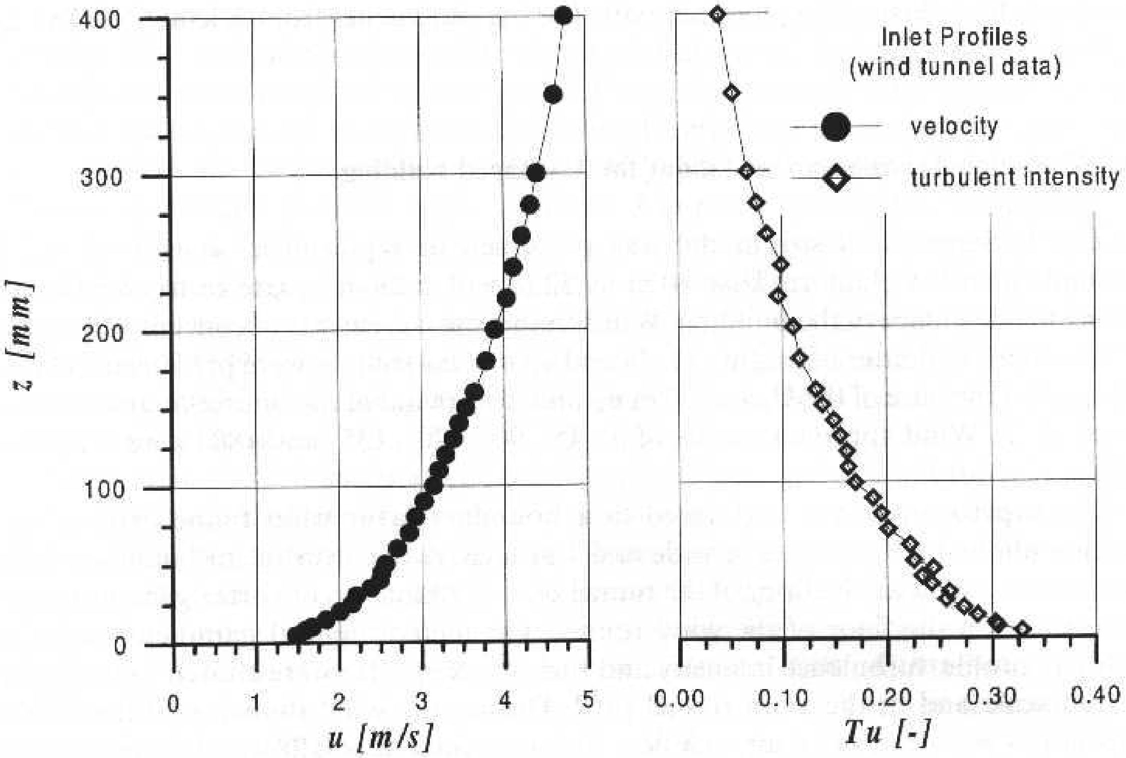

Fig. 2. Mean velocity and turbulent intensity profiles approaching U-shaped building.

4. Numerical model configuration

Version 4.3 of the FLUENT code as well as the new FLUENT/UNS Version 4.0 were used for numerical simulations. Calculations were performed with both structured and unstructured grid generation (Fig. 3). The calculation domain was a grid 94 cells high, 23 cells wide, and 29 cells long for the structured grids. A plane of symmetry through the longitudinal building center was utilized for the 0° and 180° wind orientations. For the 45° orientation the entire building was placed in an unstructured grid volume of 36 875 tetrahedral cells. Outlet and pressure inlet or symmetry boundaries were specified at the sides and top of the grid volume, while a surface roughness of 0.01 m was specified at the ground and 0 m on all the building surfaces. The roughness height was selected based on calculations with different uniform ground roughness (no building) where at 0.01 m roughness height no change in velocity and turbulent intensity profile was observed. The inflow boundary conditions were chosen to match the velocity and turbulence profiles measured during the wind-tunnel experiments (see Fig. 2). Outflow boundary conditions were chosen to maintain constant longitudinal rate of change of all dependent variables (i.e. constant slope). In order to minimize the computing time all simulations were started as laminar solutions without species transport from an initial guess based on the values given at the flow inlet. Starting from the laminar solution the turbulent flow field was calculated in the second step and the species transport was added to the solution process when the turbulent flow field was almost converged. To avoid stability problems caused by pressure outlet boundary conditions, the calculations were started with symmetry conditions at the side walls and at the top of the computational domain. When a stable result was found, the pressure outlet conditions were used for a final set of iterations to avoid artificial bounding of the flow. Several tests with a structured body-fitted rectangular grid system were also carried out for the 45° setup. Because of higher grid skewness the 45° grids tended to cause instabilities during the iteration process which produced irrelevant solutions. The optimized unstructured grid was found to be significantly more stable and robust even when changing boundary conditions. FLUENT provides a set of tools for generating and adapting unstructured triangular/tetrahedral meshes with relative ease. Thus, the total number of cells used in an unstructured grid could be optimized for solution accuracy, stability and speed.

The ReNormalized Group k–ε model was used to provide turbulent closure during the calculations. This model precludes the need to specify any calibration constants or to use wall function approximations near surface boundaries.

5. Results

The FLUENT program calculated distributions of mean velocity, turbulent energy, pressure and mean concentrations. Since the mean concentrations available from the wind-tunnel experiments were given as standardized normalized concentration, $K = Cu_{10}/Q$, with the dimensions $1/\mathrm{m}^2$ all FLUENT results were similarly normalized for comparison in the following diagrams. Laboratory and numerical model results are presented in the form of scatter diagrams, bar charts and iso-concentration profile plots.

5.1. Downwind maximum concentrations near the ground

The wind-tunnel results have been compared with numerical calculations from the ABC, ASMUS, DASIM, FLUENT and MISKAM microscale models. A comparison is provided between maximum concentrations values found near the ground in a cross-section 40 m behind the building. In Fig. 4 the calculated maximum values for all models are plotted against the wind-tunnel results for the 28 m high U-shaped building and the 3 different sources. The figure illustrates the scatter of the numerical calculations. The calculated values vary mainly in the range 5 times bigger and 5 times smaller than the corresponding wind-tunnel results, but for some calculations the ratio of numerical calculation to wind-tunnel experiment is less than one tenth. FLUENT calculations with the RNG turbulent model fall in the same range of accuracy except for source B and C at 0° wind direction and source C at 180° wind direction. FLUENT predicted significantly higher concentrations (up to a factor of 32), when a stationary vortex structure was near the 40 m profile. Otherwise the model DASIM produced the worst comparison range.

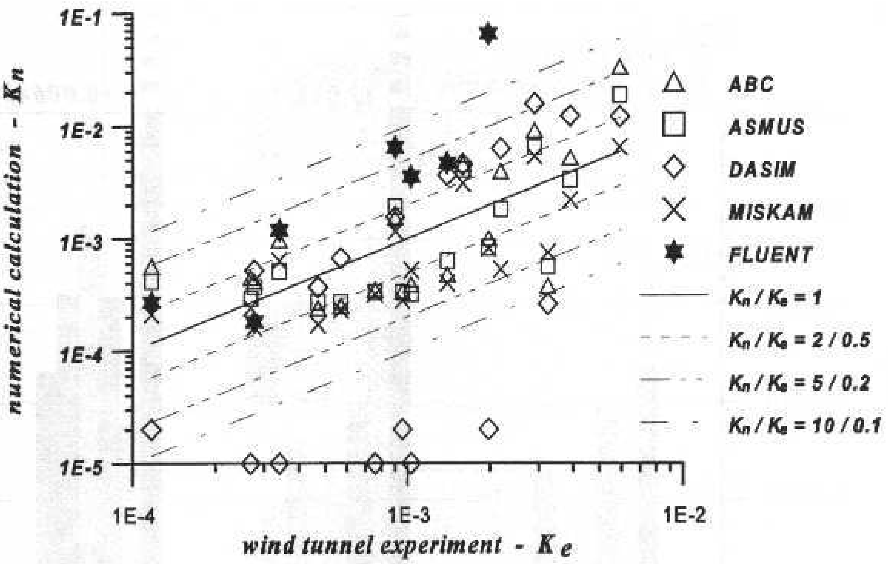

Fig. 4. Comparison between wind tunnel and numerical models using maximum concentrations near ground 40 m behind U-shaped building, height $H_b = 28$ m.

In Fig. 5 bar diagrams are shown, where calculated maximum concentrations at 40 m downstream of the building are compared with the corresponding wind-tunnel values. DASIM consistently underpredicts pollution for source A, located on the roof of the building. It also incorrectly predicts that almost no pollutants reach the ground; hence, the large discrepancies in the scatter diagram, Fig. 4. Concentrations calculated with FLUENT agree well with wind-tunnel results for the roof source situations (source A, 0°/180° wind direction) except at the 40 m downstream section. In a steady-state FLUENT simulation, stationary vortex structures are calculated at the edges of the building, and these structures are usually enriched with higher concentrations of tracer gas. In a wind tunnel as well as in nature, vortex shedding would lead to significantly higher mixing and lower local tracer concentrations. A lower lateral dispersion downstream of the building was observed for all FLUENT simulations. Once again the stationary simulation which does not lead to the characteristic vortex shedding at the building obviously caused this discrepancy, and a time dependent solution should give at least more intensive lateral mixing.

Large discrepancies were also found for most of the ground level release situations. Even if the roughness height was chosen to give similar inflow and outflow profiles for test calculations with a rough surface, it might be necessary to explicitly simulate sharp edged surface roughness elements. An artificial increase of turbulent intensity at the flow inlet as well as a significant increase of the surface roughness did not show a corresponding increase in turbulent mixing near the ground. Source B release at 180° wind direction was the only case were FLUENT predicted lower concentrations than measured in the wind tunnel.

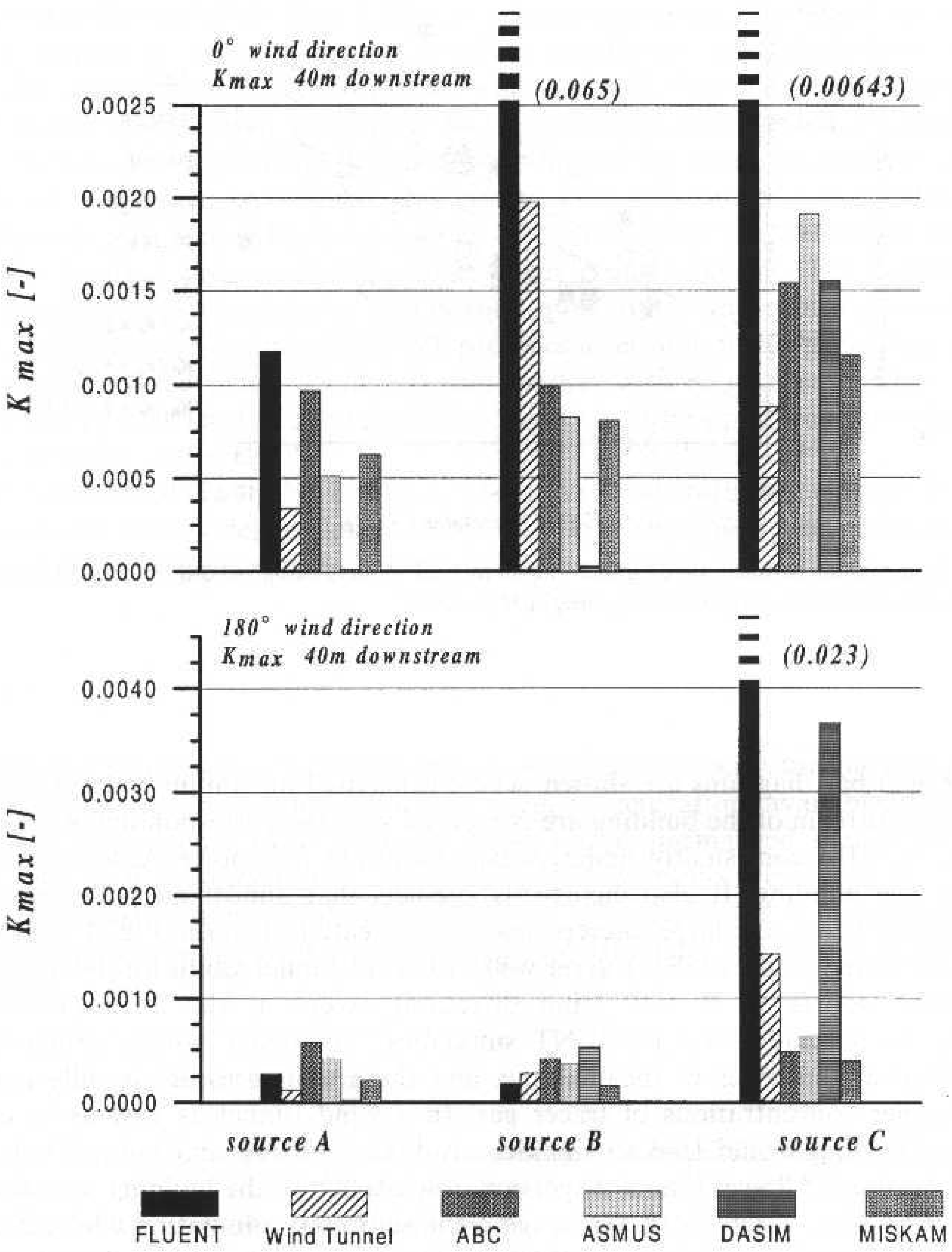

Fig. 5. Comparison between wind tunnel and numerical models using maximum normalized concentrations measured near ground 40 m behind U-shaped building, height $H_b = 28$ m.

5.2. Isolines along a symmetry axis

Isolines of concentration in a vertical plane through the building symmetry axis are shown for the case of source B and wind direction 0° in Fig. 6 (K values are multiplied by a factor of 1000). The figure demonstrates the differences between the calculated plumes. ASMUS, FLUENT and MISKAM show the best agreement with the

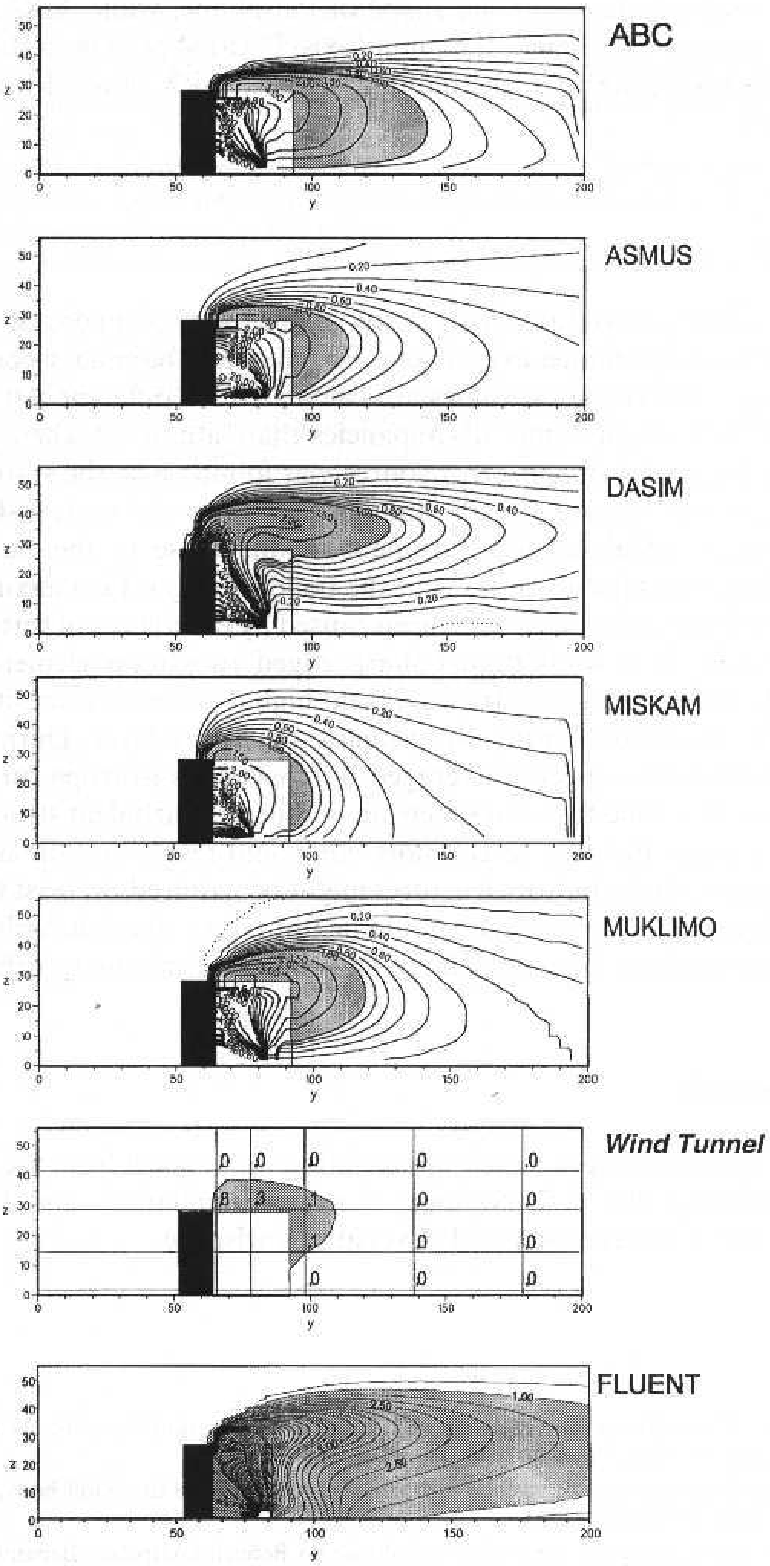

Fig. 6. Iso-concentration lines from wind tunnel and numerical models along longitudinal centerline cross-sections for the case of source B and wind direction 0°.

wind-tunnel results in terms of the shape of the plume, while ABC and FLUENT predict high concentrations near the plume axis. DASIM predicts a shift of the plume axis up to heights higher than the building height which causes lower ground level concentrations.

6. Conclusions

FLUENT gives a mixed image in terms of accuracy of predicted concentrations compared to the wind-tunnel experiment. For identical boundary conditions higher as well as lower concentration values are calculated for different test cases. Ground level sources show much higher discrepancies than situations where the tracer was emitted from the roof. A major error source was found to be the stationary solution procedure that was chosen for all simulations. Since no vortex shedding at the building edges is calculated less turbulent mixing close to the building leads to stationary high concentration areas near the building edges. Less mixing observed for ground level releases might also have been caused by differences in turbulent structure close to the wall. In a wind tunnel sharp edged roughness elements are used to simulate large scaled turbulent structures and high turbulence intensities close to the wall that meet conditions in the atmospheric boundary layer. During a stationary FLUENT simulation a spectral-averaged homogeneous-isotrope turbulence is presumed whereas in a wind tunnel a spectrum of different turbulent structures take part in turbulent mixing. In order to get more consistent results a more accurate way of simulating large-scale turbulent structures might be required. At least time-dependent solutions or large eddy simulations should be utilized to simulate high turbulent flow fields in an atmospheric boundary layer and turbulent mixing in urban areas.

Acknowledgements

The authors wish to express their appreciation for support from the Alexander von Humboldt Stiftung, the U.S. National Science Foundation, and the Institut für Hydrologie und Wasserwirtschaft, Universität Karlsruhe.

References

[1] J. Eichorn, Entwicklung und Anwendung eines dreidimensionalen mikroskaligen Stadtklima-Modelles, Dissertation, Universität Mainz, Germany (1989).

[2] S. Murakami, Comparison of various turbulence models applied to a bluff body, J. Wind Eng. Ind. Aerodyn. 46 & 47 (1993) 21–36.

[3] R. Röckle, Bestimmung der Strömungsverhältnise im Bereich komplexer Bebauungstrukturen, Dissertation, TH Darmstadt, Germany (1990).

[4] S. Blinda et al., Entwicklung und Verifizierung eines Rechenmodells zur Simulation der Ausbreitung von KFZ-Emissionen in Berich komplexer Gebäudekonfigurationen, Abschlußbericht MURL, NRW, Inst. F. Meteorogie, TH Darmstadt, Germany (1992).

[5] G. Gross et al., ASMUS – Ein numerisches Modell zur Berechnung der Strömung und der Schadstoffverteilung im Bereich einzelner Gebäude, Met. Rundschau (1994).

[6] P. Klein, M. Rau, R. Röckle, E.J. Plate, Concentration estimation around point sources located in the vicinity of U-shape buildings and in a built-up area, 2nd Int. Conf. Air Pollution, Barcelona, Spain, 27–29 September 1994.

[7] H. Panskus, Ein mikroskaliges diagnostisches Strömungs-und Ausbreitungsmodell für komplex bebautes Gelände – Evaluierung und Validation, Diplomarbeit im Fach Meteorologie, Universität Hamburg, Hamburg, Germany, (1995) 154 pp.

[8] V. Yakhot, S.A. Orzag, Renormalized group analysis of turbulence – I: basic theory, J. Sci. Comput. 1 (1986) 1–51.

[9] P. Klein, M. Rau, Z. Wang, E.J. Plate, Ermittlung des Strömungs- und Konzentrationsfeldes im Nahfeld typischer Gebäudekonfigurationen (Experimente), Paper 10, Statuskolloquium des PEF, 15–17 März 1994 und PEF-Abschlussbericht, Kernforschungszentrum Karlsruhe, Germany (1994a).

[10] E.J. Plate (1982) Wind tunnel modelling of wind effects in engineering, in: Engineering Meteorology, Ch. 13, Elsevier, Amsterdam, pp. 573–639.

[11] Fluent User's Guide, Vols. 1–4, Fluent Inc., Centerra Resource Park, Lebanon, New Hampshire, USA.