ELSEVIER

Journal of Wind Engineering
and Industrial Aerodynamics 89 (2001) 1325–1334

JOURNAL OF wind engineering AND industrial aerodynamics

www.elsevier.com/locate/jweia

Numerical and physical modeling of bluff body flow and dispersion in urban street canyons

Cheng-Hsin Chang, Robert N. Meroney*

Wind Engineering and Fluids Laboratory, Department of Civil Engineering, Colorado State University, Fort Collins, CO 80523, USA

Abstract

To develop reliable computer models for the bluff body flow and transport of pollutants or chemical and biological (CB) agents in urban environments requires accurate measurements of the basic flow fields for carefully controlled, well-known conditions. Fluid modeling in an industrial wind tunnel provides an opportunity to produce accurate simulations of the bluff body flow and transport of urban pollution or of CB agents associated with urban terrorism incidents. A basic building shape, the Wind Engineering Research Field Laboratory building (WERFL) at Texas Tech University, is used for this study. The urban street canyon was represented by a 1:50 scale WERFL model that was surrounded by models of similar dimensions. These buildings were arranged in various symmetric configurations with different separation distances and different numbers of surrounding building. A series of measurements is made over a generic urban street canyon arrangement using flow visualization, anemometry, pressure transducer and gas chromatography. The experimental data include visualization, velocity and turbulence intensity profiles, surface pressure on the building and dispersion of releasing gas. Results are compared to three-dimensional numerical models of the same configuration using the commercial code, FLUENT 5.3. The effects of grid resolution, boundary conditions, source placement and selection of turbulence model (kappa-epsilon, RNG kappa-epsilon, Reynolds stress, etc.) are examined in a series of sensitivity calculations. © 2001 Elsevier Science Ltd. All rights reserved.

Keywords: Urban street canyon; Dispersion; Bluff body flow; Numerical modeling

---

1. Introduction

The flow patterns that develop around individual buildings govern the wind forces on the building and the distribution pressure about the building and pollution about

---

*Corresponding author. Tel.: +1-970-491-8574; fax: +1-970-491-8671.
E-mail address: meroney@engr.colostate.edu (R.N. Meroney).

PII: S0167-6105(01)00129-5

CEP 00-01-3

software is based on a finite volume discretization of the equations of motion, an unstructured grid volume made of either rectangular prisms or tetrahedral cells, various matrix inverting routines, and, in this case, either kappa–epsilon ($\kappa-\varepsilon$) or renormalized group theory kappa–epsilon (RNG $\kappa-\varepsilon$) turbulence model. Steady–state solution was sought for several flow configurations and the data generated were displayed on various isopleth contour plots of velocity, turbulence and concentration. Particle trajectories were also generated to elucidate the effects of building spacing and street configurations.

2. Results

As noted previously multiple building configurations were considered (see Fig. 1 and Table 1). Depending on the street width to building height ratio (B/H), the flow in the street canyons can be classified as skimming flow ($B/H = 0-1.2$), wake interference flow ($B/H = 1.2-5.0$), or an isolated roughness flow ($B/H > 0.5$) as originally proposed by Oke [14]. Results differed substantially depending upon whether the master building was surrounded by only a few or many surrounding buildings. If the surrounding depth of buildings about the central structure were only one or two circuits deep, the approach flow was characterized by the open-country roughness surrounding the complex. But if multiple building circuits existed, then an equilibrium urban roughness situation existed.

2.1. Open-country case

Characteristics of flow, pressure and concentration distributions were examined. It was noted that:

- Visualization using smoke and vertical light sheets revealed that clean air is drawn into the canyon by intermittent eddies circulating down into the canyons or along the upwind street canyon intersections;
- Significant pollution concentrations were measured on building faces upwind of ground level sources and along rooftops and
- Stagnation pressures occurred on the upwind face of the test building, but their magnitudes were reduced by the sheltering effect of upwind model structures.

2.2. Urban roughness case

Experiments were also performed on the dispersion within extended urban roughness associated with additional up- and down-stream dummy buildings. The overall characteristics discussed for the open-country case were identified also in canyons amidst large urban roughness, but some significant differences were observed:

- For closely spaced multiple street canyons, skimming flow dominates, advection in and out of the canyons appears intermittent, and mixing over the street-canyon top streamline appears to primarily by turbulent mixing.

- As the street widths widen with respect to building height wake-interference flows dominate the advection and dispersion of pollutant plumes.
- Once the street width to building height exceeds about 5, the flow field even for a multiple building arrangement appears to be perturbed by individual isolated buildings.

2.3. Numerical simulation

Version 5.3.18 of the FLUENT code and version 1.2.0 of GAMBIT unstructured grid code were used for numerical simulations. The code was run on an AMD Athlon 750 MHz PC using a Microsoft Windows 2000 Operating System. Four separate turbulence models, standard $k-\varepsilon$, RNG $k-\varepsilon$, Reynolds-stress and LES, were examined for each case. Calculations were preformed with unstructured grid generation (Fig. 2). The line source inlet was modeled as $dx = 0.5$ cm in width and $W = 18.4$ cm in length and set as a constant velocity inlet with no turbulence. The inlet velocity of $w_{\text{source}} = 0.01$ m/s, equivalent to the source emission rate used in the wind tunnel simulation. A tracer mass fraction of 1 was applied to the line source inlet during the calculation.

The wind tunnel profiles of velocity and turbulence intensity (CSU-B2) could be used for calculating boundary conditions for an inlet profile.

Since the wind tunnel results of concentration measurements were provided in a non-dimensional form, C were normalized in the same way with source emission rate (w_{source}, dx), the height of the building, H, and reference velocity, U_{ref}:

$$K = CHU_{\text{ref}}/w_{\text{source}}\,dx. \tag{1}$$

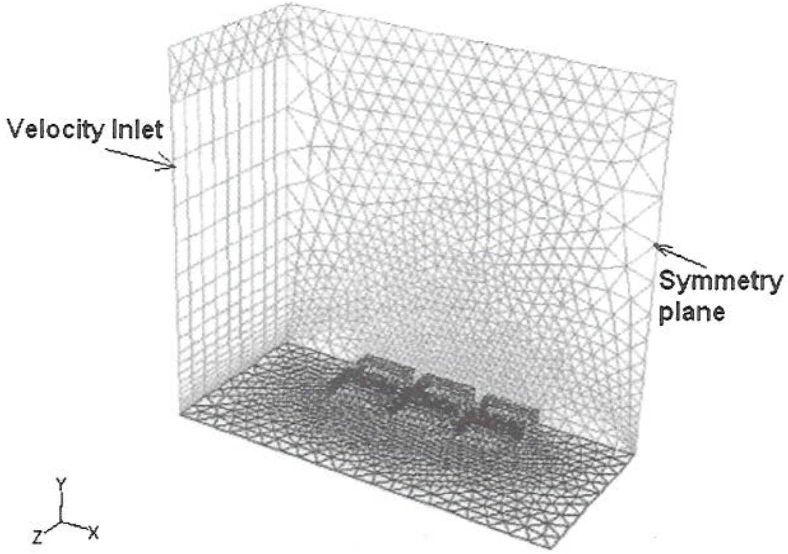

Fig. 2. Unstructured boundary mesh used for tetrahedral mesh generation.

Comparing each set of data from wind tunnel simulation and numerical simulation, most of the results showed that the CFD software, Fluent 5.3, can well predict the actual wind tunnel simulation data by choosing optimum boundary condition, grid resolution and turbulence model. Figs. 3 and 4 show the comparisons between wind tunnel measurements and calculations in both velocity and turbulence intensity profiles which were taken from the center of line source inlet. For this case, canyon width ratio $B/H = 1$, the calculated results agree well with the measurements in the wind tunnel by running $k-\varepsilon$ turbulence model.

Fig. 5 indicates the positions of concentration measured for the comparison of wind tunnel data and calculated data. Fig. 6 shows a direct comparison between measured and calculated concentrations on the central line of the upwind and downwind walls for street canyon where line source inlet located for case $B/H = 1$, $N = 1$.

Fig. 7 shows the comparison of concentrations on the central line of roof surface. The calculated concentrations (Standard $k-\varepsilon$) for the upwind wall of the street canyon and the roof surface agree well with the data collected from the wind tunnel. At the downwind wall, the calculated values were significantly higher than the values measured in the wind tunnel.

A comparison of measured and calculated concentrations for the $B/H = 1$ with different number rows of shelter models is given in Fig. 8. Two concentration taps were chosen. One was on the downwind wall and the other was on the roof surface. Both measured and calculated results showed increased concentrations by the sheltering effect of upwind model structures.

As noted previously by Meroney et al. [15], it is not difficult to achieve a "correct" looking presentation of pressures and concentrations over a bluffbody; however, it is not given that quantitative equivalence between experimental and numerical data

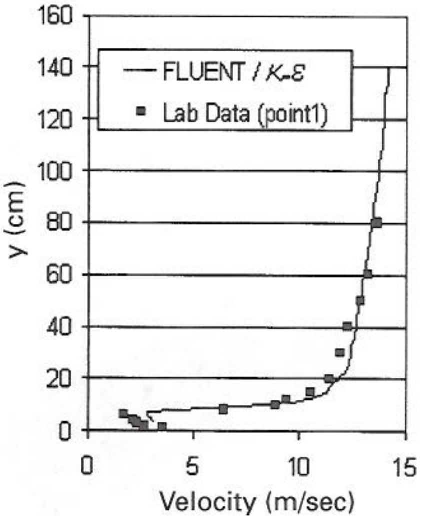

Fig. 3. Comparison of velocity profile between calculated and measured data.

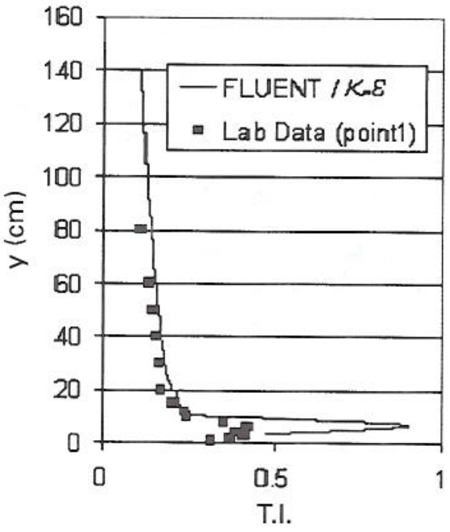

Fig. 4. Comparison of T.I. profile between calculated and measured data.

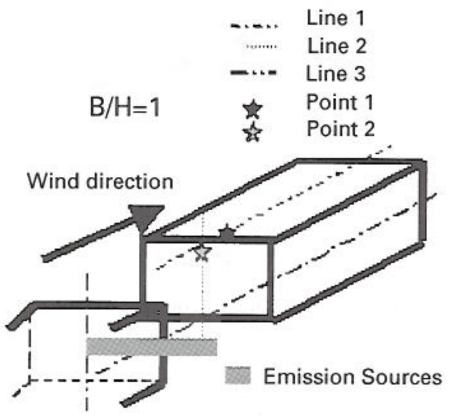

Fig. 5. Positions of concentration measured for the comparisons.

will occur unless careful attention is paid to inlet profiles, grid adaptation and the turbulent model chosen. In the calculation produced to replicate some of the test cases studied above, it was found necessary to take utmost care in adapting the turbulent grids to assure that intense concentration gradients, separation locations and re-attachment locations were reproduced.

3. Conclusion

Wind-tunnel flow, pressure and diffusion tests performed about an idealized building arrangement replicated many of features of urban environment previously

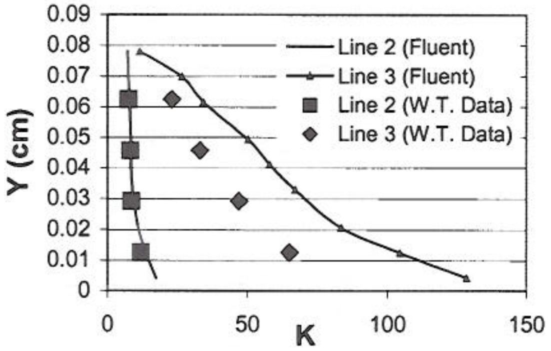

Fig. 6. Comparison between measured and calculated concentrations on the upwind and downwind walls.

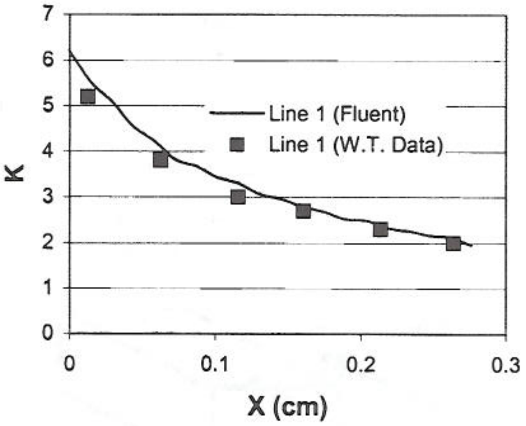

Fig. 7. Comparison between measured and calculated concentrations on the central line of roof surface.

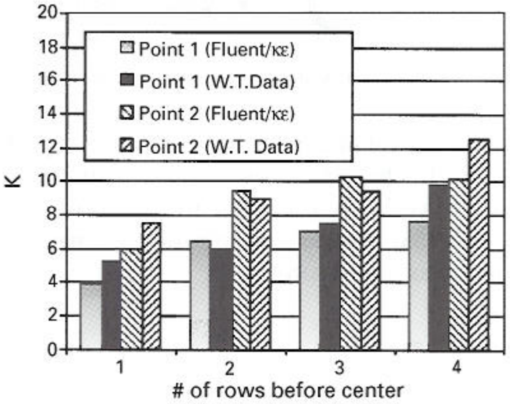

Fig. 8. Calculated and measured concentration for different N.

noted at full scale and in previous laboratory simulations. Numerical simulations using FLUENT reproduced these patterns, but only with care taken to provide adequate grid resolution, accurate inlet flow profiles, and improved turbulence models.

Conclusions of the results of the experiment with FLUENT can be summarized as follows:

- Fluent 5.3 can simulate the flow field in urban street canyons very well using $k-\varepsilon$ model,
- Standard $k-\varepsilon$ model and RNG $k-\varepsilon$ model give almost the same results,
- Predicted separation and reattachment areas agree well with results from visualization of wind tunnel simulation,
- Concentration magnitude is under-predicted by numerical models for cases of more rows of shelter buildings,
- Well setup of grid resolution gets faster numerical solutions converged and
- Adapted grids provides a convenient way to reproduce flow details of separation, reattachment, and high concentration regions without excessive calculation cells.

Acknowledgements

The contribution of Dr. David E. Neff in implementing the experimental facilities at CSU is gratefully acknowledged. This work was supported by the US National Science Foundation through the CSU/TTU Cooperative Program in Wind Engineering, Grant No. CMS-9411147.

References

[1] W. Theurer, Point sources in urban areas: modeling of neutral gas clouds with semi-empirical models, in: Cermak et al. (Eds.) Wind Climate in Cities, Kluwer Academic Publishers, Dordrecht, 1995, pp. 485–502.

[2] D. Banks, R.N. Meroney, A model of root-top surface pressure dependence upon local flow parameters, in: Larsen, Larose, Livesey, (Eds.), Wind Engineering into the 21st Century, Balkema Press, Rotterdam, 1999, pp. 1097–1104.

[3] D. Banks, R.N. Meroney, P.P. Sarkar, Z. Zhao, F. Wu, Flow visualization of conical vortices on flat roofs with simultaneous surface pressure measurement, J. Wind Eng. Ind. Aerodyn. 84 (1999) 65–85.

[4] R.N. Meroney, D.E. Neff, B. Birdsall, Wind-tunnel simulation of infiltration across permeable building envelopes: Energy and air pollution exchange rates, Seventh International Symposium On Measurement and Modeling of Environmental Flows, International Mechanical Engineers Conference, San Francisco, 12–17 November, 1995, 8pp.

[5] R.N. Meroney, S. Rafailidis, M. Pavageau, Dispersion in idealized urban street canyons, in: Gryning, Schiermeir (Eds.), Air Pollution Modeling and Its Application, XI, Plenum Press, New York, 1996, pp. 451–458.

[6] B. Leitl, R.N. Meroney, Car exhaust dispersion in a street Canyon, Numerical critique of a wind-tunnel experiment, J. Wind Eng. Ind. Aerodyn, 67&68 (1997) 293–304.

[7] S. Rafailidis, M. Pavagean, M. Schatzmann, Wind tunnel simulation of car emission dispersion in urban street canyons, in: Analen der Meteorologie, Deutsche Meteorologische Gesellschaft, Munich, 1995.

[8] B. Leitl, P. Kastner-Klein, M. Rau, R.N. Meroney, Concentration and flow distributions in the vicinity of U-shaped buildings: Wind-tunnel and computational data, J. Wind Eng. Ind. Aerodyn. 67&68 (1997) 745k–755k.

[9] M. Rau Klein, R. Rockle, E.J. Plate, Concentration estimation around point sources located in the vicinity of U-shaped buildings and in a built-up area, Second International Conference Air Pollution, Barcelona, Spain, 27–29 September 1994.

[10] B. Bienkiewicz, First five years of NSF's collaborative program in Wind Engineering: results and future activities, 27th Joint UJNR Meeting of the US-Japan Panel on Wind and Seismic Effects, 16–19 May, 1995, Tsukuba, Japan, 23pp.

[11] J.B. Birdsall, Physical simulation of wind-forced natural ventilation, Master Thesis, Department of Civil Engineering, Colorado State University, 1993.

[12] L.S. Cochrane, Wind tunnel Modeling of low-rise structures, Ph.D. Dissertation, Civil Engineering, Colorado State University, 1992, 348pp.

[13] H.J. Ham, Turbulence effects on wind-induced building pressures, Ph.D. Dissertation, Civil Engineering, Colorado State University, 1998, 275pp.

[14] T.R. Oke, Street Design and Urban Canopy Layer Climate, Energy and Buildings 11 (1998) 103–113.

[15] R.N. Meroney, B.M. Leitl, S. Rafailidis, M. Schatzmann, Wind-tunnel and numerical modeling of flow and dispersion about several building shapes, J. Wind Eng. Ind. Aerodyn 81 (1998) 333–345.