Preprint
UCRL-JC-142234

Validation of Two CFD Urban Dispersion Model using High Resolution Wind Tunnel Data

S.T. Chan, S.E. Stevens

This article was submitted to 3rd International Symposium on Environmental Hydraulics Conference, Tempe, AZ, December 5-8, 2001

U.S. Department of Energy

Lawrence
Livermore
National
Laboratory

July 13, 2001

This work was performed under the auspices of the United States Department of Energy by the University of California, Lawrence Livermore National Laboratory under contract No. W-7405-Eng-48.

Validation of Two CFD Urban Dispersion Models Using High Resolution Wind Tunnel Data

S.T. Chan, D. E. Stevens
Lawrence Livermore National Laboratory
and
W.S. Smith
Los Alamos National Laboratory

INTRODUCTION

Numerical modeling of air flow and pollutant dispersion around buildings in the urban environment is a challenging task due to the geometrical variations of buildings and the extremely complex flow created by such surface-mounted obstacles. Building-scale air flows inevitably involve flow impingement, stagnation, separation, a multiple vortex system, and jetting effects in street canyons. Lawrence Livermore National Laboratory (LLNL) and Los Alamos National Laboratory (LANL) have developed two complementary, robust computational fluid dynamics (CFD) models, FEM3MP by LLNL and HIGRAD by LANL, for such purposes. Our primary goal is to support emergency response planning, vulnerability analysis, and development of mitigation strategies for chem-bio agents released in the urban environment.

Model validation is vitally important in establishing the credibility of CFD models. We have, in the past, performed model validation studies involving simpler geometries, such as flow and dispersion past a cubical building [1] and flow around a 2-D building array [2]. In this study, wind tunnel data for a 7x11 array of cubical buildings [3] are used to further validate our models.

NUMERICAL MODELS

Our models are based on solving the three-dimensional, time-dependent Navier-Stokes equations on massively parallel platforms, with FEM3MP focusing on the building scale and HIGRAD on the urban scale. Both models have physics submodels for aerosols, UV radiation decay, surface energy budget, and tree canopy effects. Advanced turbulence submodels are also available in both codes. In addition, FEM3MP employs the finite element method for accurate representation of complex building shapes and a linearlized, fully implicit projection method for efficient time-integration, whereas HIGRAD utilizes a terrain-following coordinate system and finite difference techniques to solve the equations.

FEM3MP has a nonlinear eddy viscosity (NEV) and a Smagorinsky large eddy simulation (LES) turbulence submodels. The NEV turbulence submodel [4, 5] has many desirable properties, including anisotropy, a cubic constitutive law, and no need for wall functions. Such a submodel, coupled with a linearized, implicit projection method, has made the numerical algorithm highly cost-effective for simulating flows and dispersion around buildings.

HIGRAD is a large-eddy CFD code that is second order accurate in space and time. The model uses a non-oscillatory forward in time advection scheme that can accurately model regions of strong shear[6]. It also uses an efficient conjugate residual pressure solver , and either a Smagorinsky or a single equation TKE based [7] subgrid closure. A simple law-of-wall parameterization is applied near the building surfaces.

WIND TUNNEL EXPERIMENTS

The experimental data used in this paper were from a recent USEPA wind tunnel study of flow and dispersion around a 7x11 array of cubical model buildings [3]. The cubical blocks, with height (H) of 15 cm, were arranged 15 cm apart and with 7 blocks in the windward and 11 blocks in the lateral directions. A neutral atmospheric boundary layer, with a mean wind speed of U=3 m/s at the building height, was simulated in the wind tunnel using spires and floor roughness elements upstream of the buildings. High resolution measurements of the velocity components were taken at various heights within each canyon, above model buildings, and upstream and downstream of the building array. The tracer used in the study was emitted at a rate of Q=1000 cc/min from a perforated plastic sphere placed on the ground and behind the first building. Concentration measurements were made at thirteen vertical profile positions and six lateral profile positions inside the building array. Sample results for the measured mean wind and TKE fields on the symmetry plane near the first three blocks are shown in Fig. 1.

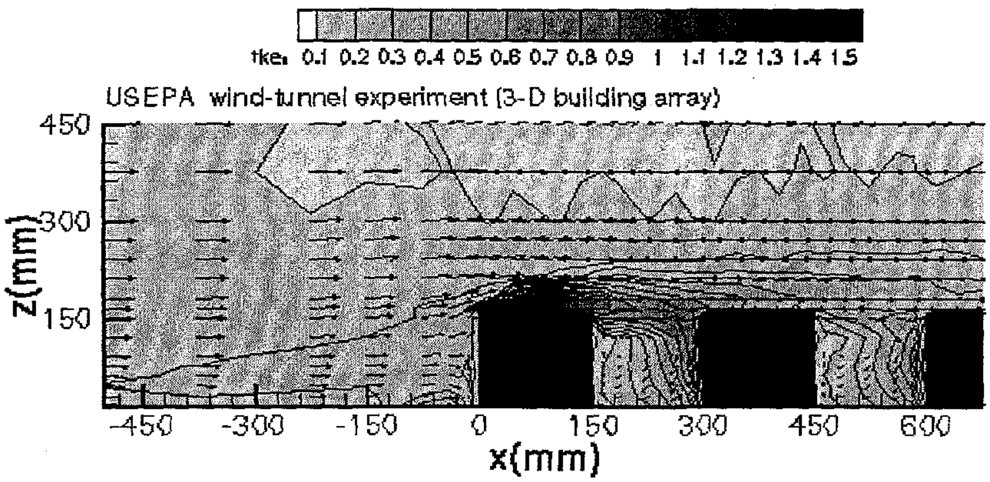

Figure 1. Wind tunnel results of mean wind and TKE fields on the symmetry plane

MODEL-DATA COMPARISON

In the following, results obtained with the NEV turbulence submodel of the Reynolds-averaged Navier-Stokes approach (referred below as RANS) in FEM3MP and the LES approach in HIGRAD are presented and compared with the experimental data.

In the RANS simulations, a computational domain of 3.6m x 1.85m x 0.6m was used, with the assumption that both the flow and dispersion patterns are symmetric about the symmetry plane of the experimental setup. Such an assumption allows us to use a relatively small, graded mesh of 169 x 121 x 25 grid points, with its finest grid resolution of 0.004 m near the building surfaces. A steady flow field was obtained after 3 s of simulation time and such a flow field was then used in the dispersion simulation for a duration of 6 s. The simulations were performed on a cluster of DEC Alpha machines. The flow simulation used 48 CPUs and took 1,500 time steps with ~15 s/step, while the dispersion simulation used 16 CPUs and took 1,200 time steps with ~4 s/step.

In Fig. 2, velocity vectors and TKE on the symmetry plane are shown. The predicted flow has a small eddy near the first building, a stagnation point at about 0.75H high in the front and a weak separated flow above the first building. Recirculation patterns in the canyons are essentially identical, with the center of eddies being about 0.8H above the ground. The TKE patterns indicate its intensity is highest near the front edge of the first building rooftop and decreases gradually in the windward direction. All these features are very consistent with the experimental results shown in Fig. 1. A more detailed model-data comparison for the velocity and TKE profiles at four representative locations are shown in Figs. 3 and 4. Despite some under-prediction of the reverse flow and TKE in the canyons, the overall agreement is very good. The predicted reattachment length behind the last building (not shown) is about 1.5H, as compared with the measured value of 1.2H.

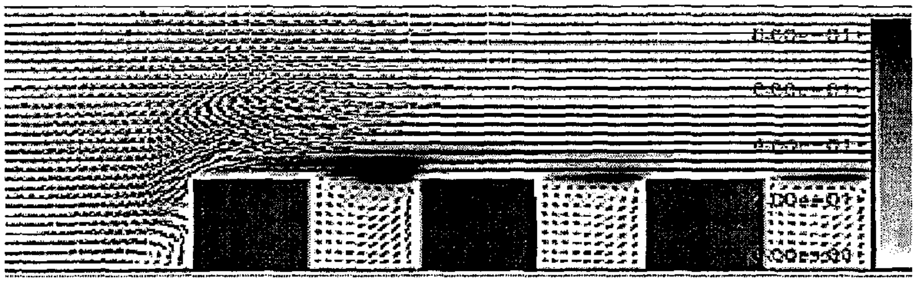

Figure 2. RANS model results of velocity and TKE on the symmetry plane near the first three blocks.

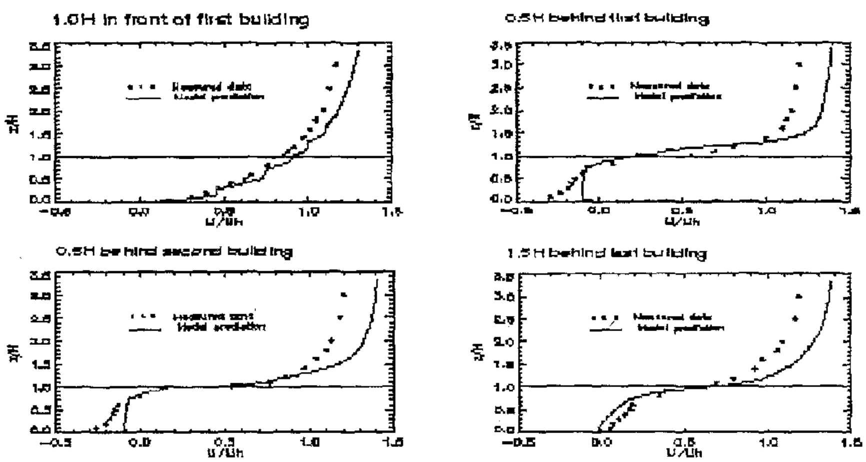

Figure 3. RANS model results of longitudinal velocity profiles versus USEPA data.

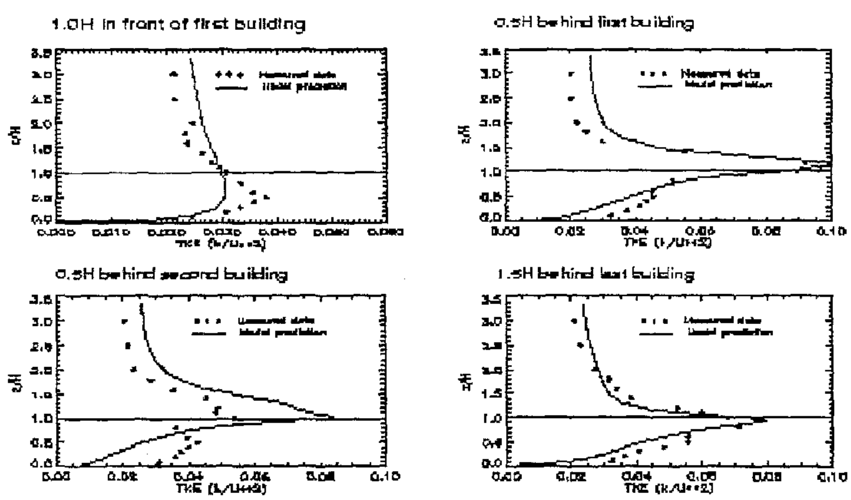

Figure 4. RANS model results of TKE profiles versus USEPA data.

Normalized concentration ($\chi = C \cdot U \cdot H^2 / Q$) on the ground and the symmetry plane are shown in Figs. 5 and 6. The predicted and measured concentrations along the centerline are also compared in Fig. 6. The vertical and horizontal extent of the plume is very consistent with the measured results (not shown) and, except for an over-prediction in the first canyon where a more sophisticated source submodel is probably required, excellent agreement is observed regarding downwind concentration.

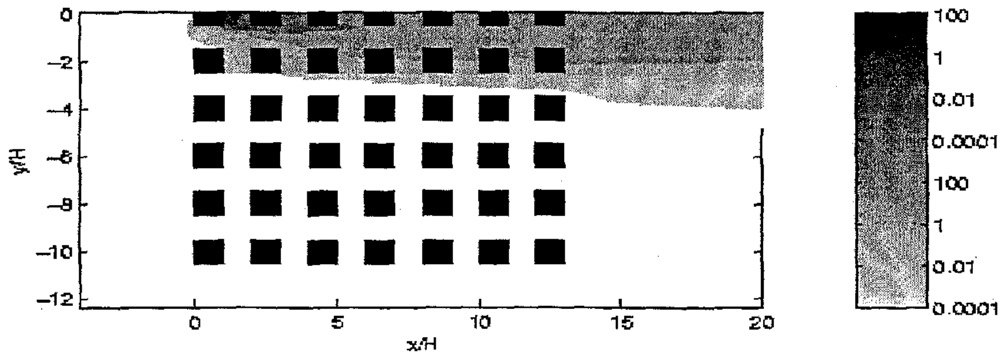

Figure 5. RANS model results of normalized concentration on the ground surface.

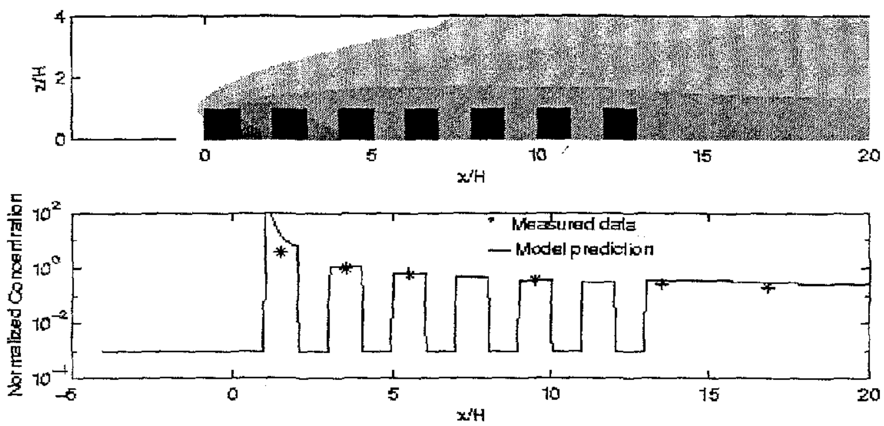

Figure 6. RANS results of normalized concentration on the symmetry plane and RANS results versus USEPA data along ground centerline.

The LES simulation was conducted using a 420x140x63 grid for a domain size of 3.15m x 2.1m x 2.1m. A 7x7 array of blocks was modeled in order to reduce the overall computational load. The resolution in the longitudinal direction was 0.0075 m and 0.015 m in the lateral direction. The vertical resolution varied with height, with the finest resolution of 0.007 m near the model surface and top of the buildings. The inflow profiles for the mean velocity and TKE were determined from the USEPA wind-tunnel data. The inflow turbulence was generated using a method similar to that used by Ahmadi and Li [8]. A time step of about 8 ms was used in order to satisfy the Courant condition. The simulation was allowed to progress for 7 s and statistics for mean velocity components and TKE were collected over the last 6 s of simulation.

Fig. 7. shows a vertical cross section of the modeled TKE and mean velocity vectors for the first two canyons. Figures 8 and 9 show a detailed comparison of the predicted versus measured mean wind and TKE profiles at various locations. The overall features of the modeled flow agree fairly well with the wind-tunnel observations shown in Fig. 1. The model correctly predicts the mean recirculation in the canyons and the recirculation over the first building. The model also correctly predicts the stagnation point at a height of about 0.75H. The TKE is nicely reproduced over the first building, but is underestimated in the canyons. This may be due to the length scale of the inflow TKE being too small. The overall TKE is better simulated downstream of the building array since the buildings play a dominant role in generating the TKE.

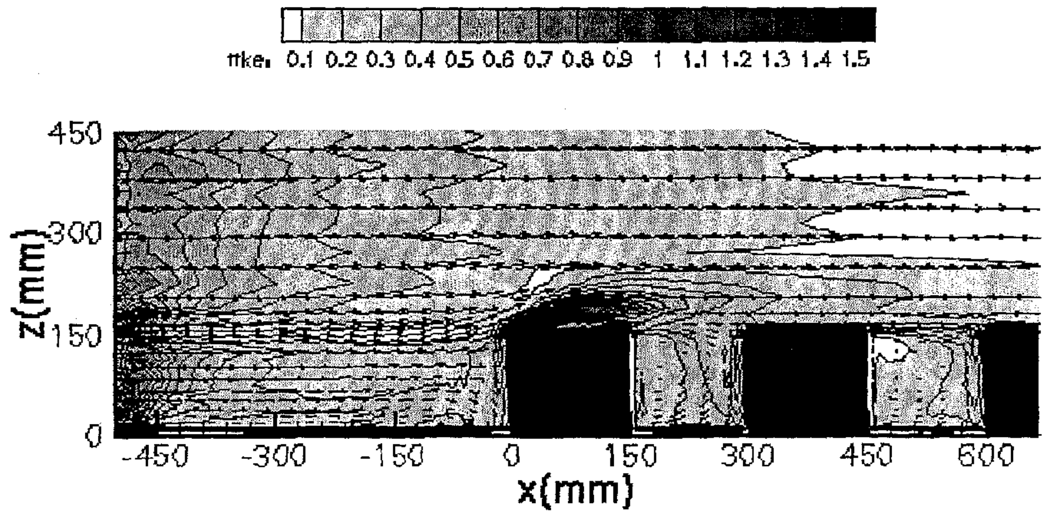

Figure 7. LES results of mean wind and TKE fields near the first two canyons.

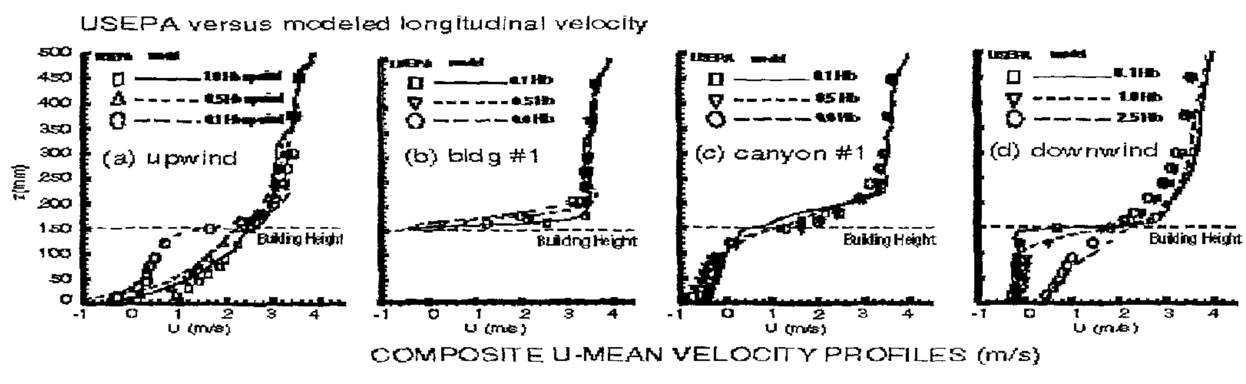

Figure 8. LES mean longitudinal velocity profiles versus USEPA data

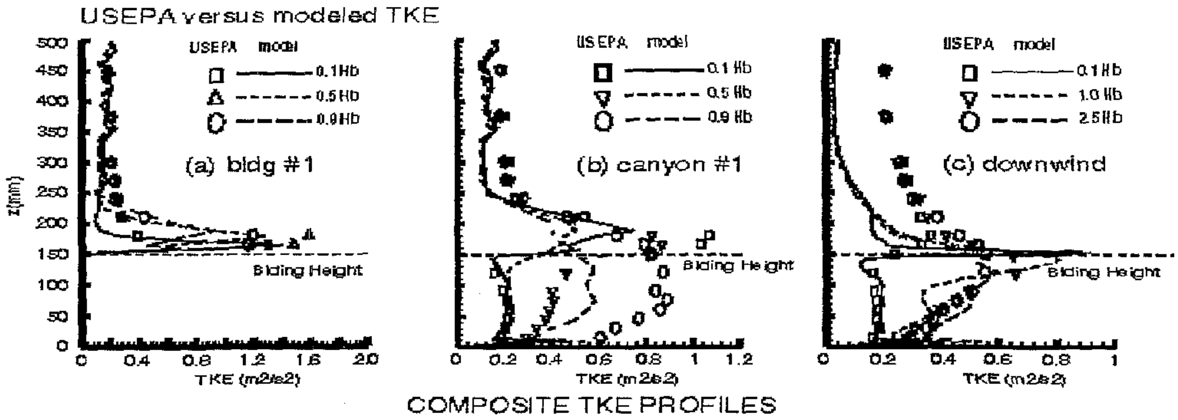

Figure 9. LES modeled TKE profiles versus USEPA data.

SUMMARY

We have presented a validation of our RANS and LES models using the flow and dispersion data obtained for a 3-D array of model buildings. Both our models are able to reproduce the important features of the experimental results, including velocity, TKE, and the concentration fields. Predicted flow field results seem to suggest that the LES results are somewhat more accurate, but at a much higher computing cost. For dispersion simulations, if mean velocity fields are adequate, the RANS approach is apparently more cost-effective. However, since an LES model can capture turbulent variations more accurately than RANS models, it is more useful for situations where accurate concentrations in both space and time are important.

REFERENCES

1. Chan, S. and D. Stevens, 2000: An evaluation of two advanced turbulence models for simulating the flow and dispersion around buildings. The Millennium NATO/CCMS Int. Tech. Meeting on Air Pollution Modeling and its Application, Boulder, CO, 355-362.
2. Smith, W., J. Reisner, D. Decroix, M. Brown, R. Lee, S. Chan, and D. Stevens, 2000: A CFD intercomparison and validation using high resolution wind tunnel data. 11th Joint AMS/AWMA Conf. on Air Pollution Meteorology, Long Beach, CA.
3. Lawson, R., S. Perry, and R. Thompson 2000: Measurement of velocity and concentration fields in arrays of 2-dimensional and 3-dimensional buildings in a simulated neutrally-buoyant atmospheric boundary layer. EPA Data Report, NOAA, Research Triangle Park, NC.
4. Suga, K., 1995: Development and application of a nonlinear eddy visibility model sensitized to stress and strain invariants, Ph.D. Thesis, UMIST, Manchester, England.
5. Gresho, P. and S. Chan: Projection 2 goes turbulent - and fully implicit. International J. of Comp. Fluid Dyn., 9, 249-272.
6. Smolarkiewicz, P. K. and L. G. Margolin, 1998: MPDATA: a finite-difference solver for geophysical flows. *J. Comp. Phys.*, **140**, 459-480.
7. Sorbjan, Z. 1996: Numerical study of penetrative and "solid lid" nonpenetrative convective boundary layers. *J. Atmos. Sci.*, **53**, 101-112.
8. Ahmadi, G. and A. Li, 2000: Computer simulation of particle transport and deposition near a small isolated building. *J. Wind Eng. Ind. Aerodyn.*, **84**, 23-46.

ACKNOWLEDGMENTS
This work was performed under the auspices of the U.S. Department of Energy by Lawrence Livermore National Laboratory and Los Alamos National Laboratory under contract number W-7405-ENG-48.