

USING COMPUTATIONAL FLUID DYNAMICS TO MODEL THE ATMOSPHERIC DISPERSION OF FIRE PLUMES

Andrew C. Ward and Ian. T. Cameron
Computer Aided Process Engineering Centre
Department of Chemical Engineering
The University of Queensland
Brisbane, Queensland
Australia

ABSTRACT

As the preliminary part of a project on modelling the atmospheric dispersion of toxic plumes from warehouse fires, the methods of computational fluid dynamics were applied to the calculation of incompressible, turbulent flow around obstacles in simulated, atmospheric boundary layers. Predictions were made for three wind tunnel studies - isothermal flow over a two-dimensional thin wall and a three-dimensional surface-mounted cube, and buoyant heated flow from a three-dimensional warehouse. Turbulence was modelled by a two-equation ($k-\epsilon$) model. The predicted flow features showed encouraging agreement with the experiments.

INTRODUCTION

Chemical and industrial accidents can pose a major hazard to people and the environment. Storage facilities such as warehouses may contain substances with flammable or toxic properties. A warehouse fire can release toxic gases into the atmosphere, and regulatory, planning, or emergency services authorities may wish to limit or determine the consequences of such an accident, by means of planning or control.

This research aims to model the near-field, atmospheric dispersion of toxic combustion products generated in warehouse fires. The results could be used by the above authorities for risk assessment and emergency response purposes. We eventually hope to obtain a parameterised representation of the modelling results for simple, fast decision-making. This paper reports the results of a preliminary investigation into the suitability of numerical modelling for predicting atmospheric flows and heated releases from buildings.

BACKGROUND AND MODELLING APPROACHES

The most common approach to predicting pollutant dispersion in the atmosphere is the use of Gaussian dispersion models. Such models apply in steady state conditions and in homogeneous, stationary turbulence (Zannetti, 1990). These types of models cannot be applied with much validity or confidence to a fire plume, in which building and inhomogeneous source effects are important. Hence, rather than attempting an empirical modification of existing models to warehouse fires, a more fundamental approach was taken: computational fluid dynamics (CFD). CFD involves finding the numerical solution to the governing, partial differential equations for the conservation of mass, momentum, energy, and species, subject to relevant boundary and initial conditions. The instantaneous equations are time-averaged and closed with a suitable turbulence formulation. The modelled equations are then made discrete and solved by iteration. For a fuller description of turbulence modelling and CFD see Rodi (1980) and Wilcox (1993). This study uses FIDAP 7.5, a commercially-available, finite element code (see Engleman, 1993).

Given our particular interest in the downwind fate of the pollutants and the research by others into modelling combustion and within-enclosure processes (Cox and Kumar, 1987), we chose to focus on the atmospheric dispersion of the fire plume. This means treating the building's roof as a flux source (Miles *et al.*, 1994). We are especially interested in the effects on the ground-level concentration caused by changes in building orientation, types and numbers of roof openings, upstream influences, and heat and mass release rates.

The steady-state, incompressible, time-averaged equations were solved. Experimental results provided

inlet conditions for the wind speed and the turbulence parameters. FIDAP applies wall functions (see Engleman, 1993) for the turbulent parameters at solid surfaces. All velocity components were set to zero at these surfaces. Elsewhere all normal derivatives were set to zero. We used the standard $k-\epsilon$ model (Wilcox, 1993) with additional terms included for buoyant flows (Engleman, 1993).

CFD VALIDATION

To assess FIDAP's ability to simulate atmospheric flows, it was tested on some documented two- and three-dimensional studies. These were the wind tunnel experiments of Perera (1981) and Castro and Robins (1977) who used pulsed-wire anemometers to measure the wind speed near a two-dimensional, thin wall and a surface-mounted cube, respectively, in a simulated atmospheric boundary layer. Both studies produced profiles of the wind speed at several points in the flow, enabling easy comparison with the numerical predictions.

In the first study, the wall was 40mm high and fitting a logarithmic wind profile to the experimental results yielded a roughness length of 0.34mm. The computational domain in FIDAP was 45 wall heights long and 16 high. The wall was assumed to have negligible thickness and was 9 wall heights from the inlet. There were 8800 quadrilateral elements and the first node was 0.1 wall heights from the solid boundaries. In the second study, the cube was 200mm high, and the fitted logarithmic profile had a roughness length of about 11mm. The computational domain was symmetric and was 22 cube heights long, 10 wide, and 10 high. There were about 27000 brick-shaped elements, and the first node was 0.02 cube heights from the solid boundaries.

We also did a preliminary investigation of FIDAP's ability to simulate the effect of buoyancy on the flow. This was based on the wind tunnel study of Hall and Waters (1986) who examined the behaviour of a buoyant gas released from the face of a building for a range of heat emission rates. In this 1:300 scale study, the building, placed across the wind direction, was 330mm long, 170mm wide, and 170mm high. The fitted roughness length was 0.63mm. The computational domain was $200h$ long, $40h$ wide, and $40h$ high, where h is the obstacle height. The building was $20h$ downwind of the inlet. There were about 27000 elements and the first node was $0.02h$ from the ground and from the building's walls. We calculated the dispersion of gas from the roof of the warehouse, assuming the gas had the same properties as air. Hall and Waters (1986) used helium, emitted at very low velocity, to simulate heated, buoyant releases ranging from 28 to 92000 W m⁻² (full-scale).

We found that numerical simulations of bluff body flows included the combined effects of growing boundary layers (see Landau and Lifshitz, 1989, for example) and the presence of obstacles. Therefore, since many wind tunnel studies report only a single reference profile relatively close to the obstacle (like Castro and Robins,

1977) or in the absence of the obstacle (like Perera, 1981), it was often impossible to reproduce numerically the conditions of such experiments. The resulting deviations between predictions and observations are likely to increase with downwind distance. To minimise these errors, all wind speeds for the wall and cube simulations were normalised by profiles calculated without the obstacles in the flow. This ensured that the effects of non-equilibrium boundary layers and different reference profiles were as small as possible. All computations were done on an SGI PowerChallenge at The University of Queensland. The flow fields proved largely insensitive to mesh density.

Some plots are shown below for these scenarios. Figures 1-3 compare numerical predictions of the thin-wall study with Perera's (1981) measurements. Vertical profiles of the mean longitudinal wind speed are plotted at distances $1.25h$ upwind, and $1.25h$, and $5h$ downwind of the wall. The agreement is generally good, and the results compare well with other similar simulations (for example, Liston *et al.*, 1993). Further downwind, the agreement is less good, as would be expected. Figures 4-8 compare the computational and experimental results for the surface-mounted cube. Again, the model seems to have captured the main features of the flow. The differences for both studies may arise for a number of reasons: as mentioned above, the lack of equilibrium boundary layers, either numerically or experimentally, and the associated difficulties in reproducing experimental conditions; measurement errors; interpolation errors (since the values for comparison were extracted from graphs in the literature); and inadequacies in the $k-\epsilon$ model for predicting some aspects of bluff body flows (Murakami, 1993).

Figure 9 presents the ground-level concentration (presented as a mass fraction) downwind of the wind tunnel building. It is clear that as the heat flux from the roof increased, less of the plume was trapped in the building's wake and so the resulting concentrations were lower. The hottest releases were able to clear the start of the wake, but some of the plume was entrained further downwind. The behaviour of the numerical simulation is very typical of that for buoyant, elevated releases (see Venkatram and Wyngaard, 1988).

CONCLUSIONS

We have shown that a CFD approach was successful in predicting the main features of the flow around bluff bodies in an atmospheric boundary layer and of a heated discharge from a building's roof. Our current work is to examine a real warehouse scenario, including exploring the effects of different flow and building conditions. We will use the results of microscale combustion experiments (Smith-Hansen and Jorgensen, 1992) to impose species boundary conditions for the flow. The final goal is to provide a simplified approach to help regulatory, planning, or emergency services personnel determine the likely impact of a warehouse fire.

REFERENCES

- Castro, I. P., and Robins, A. G., 1977, "The flow around a surface mounted cube in uniform and turbulent streams," *J. Fluid Mech.*, Vol. 79, pp.307-335.
- Cox, G., and Kumar, S., 1987, "Field modelling of fire in forced ventilated enclosures," *Combust. Sci. Technol.*, Vol. 52, pp.7-23.
- Engleman, M. S., 1993, "FIDAP 7.0 Theory Manual," FDI Inc., 500 Davis Street, Evanston IL 60201, USA.
- Hall, D. J., and Waters, R. A., 1986, "Further Experiments on a Buoyant Emission From a Building," Report No. LR567(PA), Warren Spring Laboratory, Gunners Wood Road, Stevenage, Herts SG1 2BX, UK.
- Landau, L.D., and Lifshitz, E.M., 1989, "Fluid Mechanics (2nd edition)", Pergamon Books, Oxford, 539pp.
- Liston, G. E., Brown, R. L., and Dent, J., 1993, "Application of the E - e turbulence closure model to separated atmospheric surface-layer flow," *Bound.-Layer Meteorol.*, Vol. 66, pp.281-301.
- Miles, S. D., Cox, G., Christolis, M.N., Christidou, C. A., Boudouvis, A.G., and Markatos, N.C., 1994, "Modelling the environmental consequences of fires in warehouses," *Proceedings, 4th International Fire Safety Science Symposium*, T. Kashiwagi, ed., International Association for Fire Safety Science, 1221-1232.
- Murakami, S., 1993, "Comparison of various turbulence models applied to a bluff body," *J. Wind Eng. Indust. Aerodyn.*, Vol. 46-47, pp.21-36.
- Perera, M. D. A. E. S., 1981, "Shelter behind two-dimensional solid and porous fences," *J. Wind Eng. Indust. Aerodyn.*, Vol. 8, pp.93-104.
- Rodi, W., 1980, "Turbulence Models and Their Application in Hydraulics: A State of the Art Review," International Association for Hydraulics Research, Delft, The Netherlands, 104pp.
- Smith-Hansen, L., and Jorgensen, K.H., 1992, "Combustion of Chemical Substances and the Impact on the Environment of the Fire Products," Report No. Riso-R-651(EN), Riso National Laboratory, PO Box 49, DK-4000 Roskilde, Denmark.
- Venkatram, A., and Wyngaard, J.C., eds., 1988, "Lectures on Air Pollution Modeling," Am. Meteorol. Soc., Boston, 390pp.
- Wilcox, D., 1993, "Turbulence for CFD," DCW Industries Inc., La Canada, California, 460pp.
- Zannetti, P., 1990, "Air Pollution Modeling - Theories, Computational Methods and Available Software," Van Nostrand Reinhold, New York, 444pp.

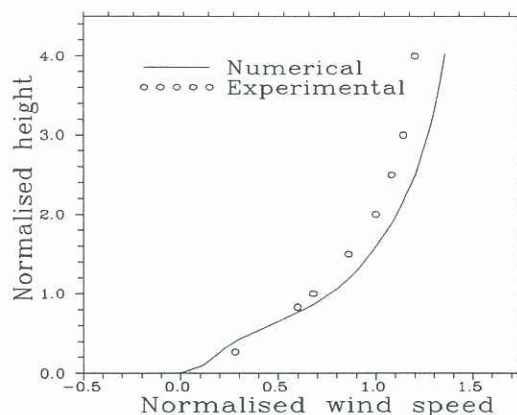


FIGURE 1. MODELLED AND MEASURED NORMALISED MEAN VELOCITIES 1.25 WALL HEIGHTS UPWIND OF THE WALL.

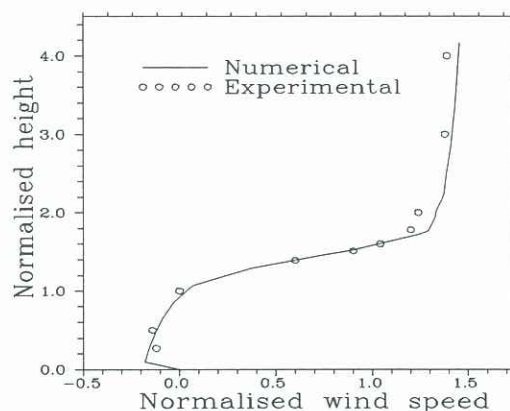


FIGURE 2. MODELLED AND MEASURED NORMALISED MEAN VELOCITIES 1.25 WALL HEIGHTS DOWNWIND OF THE WALL.

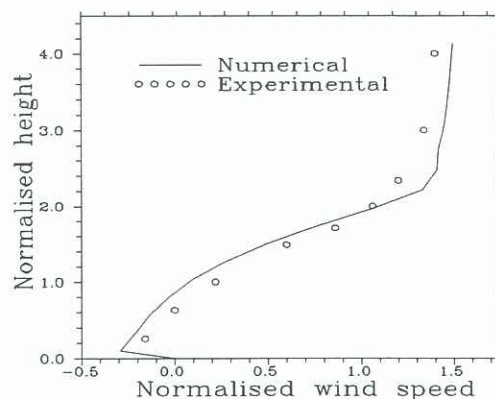


FIGURE 3. MODELLED AND MEASURED NORMALISED MEAN VELOCITIES 5 WALL HEIGHTS DOWNWIND OF THE WALL.

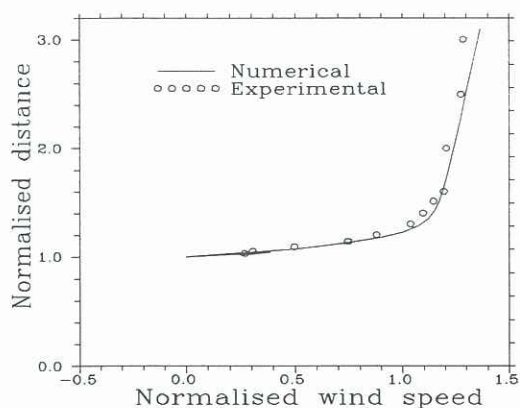


FIGURE 4. MODELLED AND MEASURED NORMALISED VERTICAL MEAN VELOCITIES ABOVE THE CENTRE OF THE CUBE.

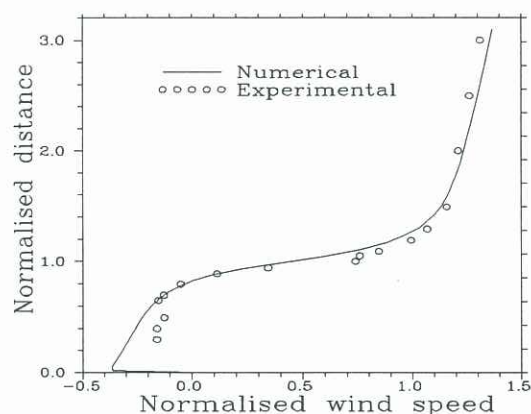


FIGURE 5. MODELLED AND MEASURED NORMALISED VERTICAL MEAN VELOCITIES 1 CUBE HEIGHT DOWNWIND OF THE CUBE.

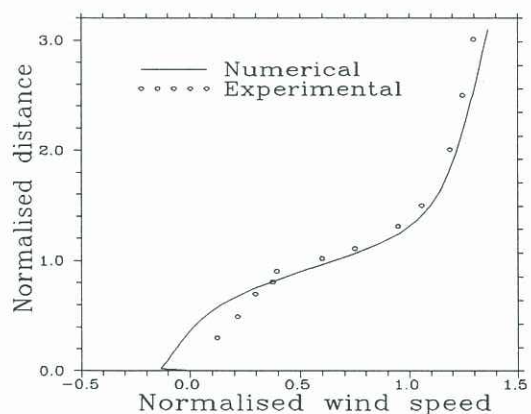


FIGURE 6. MODELLED AND MEASURED NORMALISED VERTICAL MEAN VELOCITIES TWO CUBE HEIGHTS DOWNWIND OF THE CUBE.

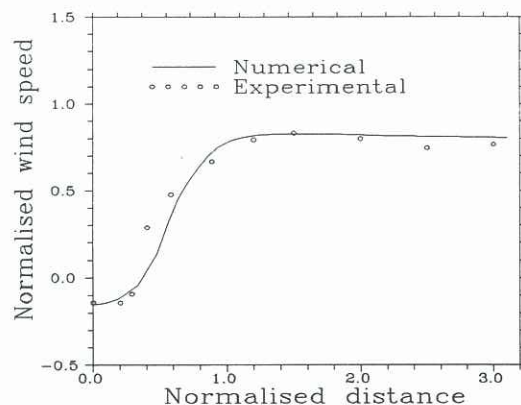


FIGURE 7. MODELLED AND MEASURED NORMALISED SPANWISE MEAN VELOCITIES 0.75 CUBE HEIGHTS DOWNWIND OF THE CUBE AND 0.5 CUBE HEIGHTS ABOVE THE GROUND.

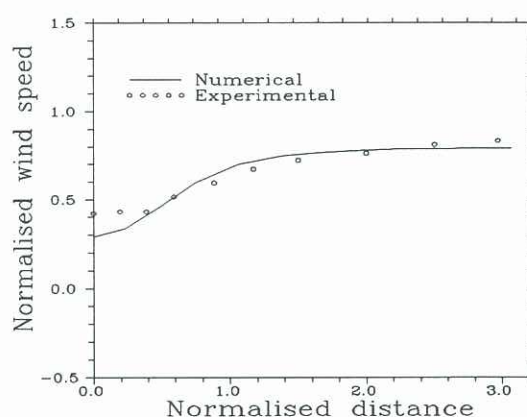


FIGURE 8. MODELLED AND MEASURED NORMALISED SPANWISE MEAN VELOCITIES 3 CUBE HEIGHTS DOWNWIND OF THE CUBE AND 0.5 CUBE HEIGHTS ABOVE THE GROUND.

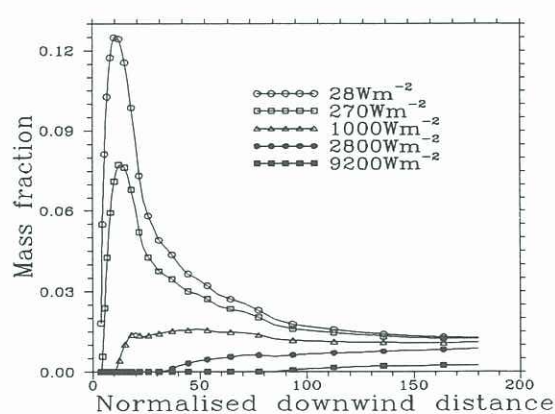


FIGURE 9. DIMENSIONLESS GROUNDLEVEL CONCENTRATIONS DOWNWIND OF ROOF SOURCES WITH DIFFERENT BUOYANCIES.