Transactions on Ecology and the Environment vol 6, © 1995 WIT Press, www.witpress.com, ISSN 1743-3541

Modelling pollutants dispersion around buildings on fire

M.N. Christolis, C.A. Christidou, A.G. Boudouvis, N.C. Markatos
Department of Chemical Engineering, National Technical University of Athens, Zografou Campus, GR-15780 Athens, Greece

Abstract

A k-ε turbulence model that accounts for buoyancy effects is used to predict flow and concentration fields around a warehouse on fire. A parametric study of the effect of the buoyancy and momentum flux number was made to examine the influence of wind speed on plume lift-off and consequently, on ground clearing. It was found that lift-off occurs at $F/U^3L=0.1$ in agreement with the values reported in the literature. The parametric analysis shows the significance of the effect of the inlet boundary conditions on flow and concentration fields.

1. Introduction

Computations of flow around buildings on fire is a very difficult task, mainly due to the presence of buoyancy and recirculation. The question of whether the plume will touch the ground or not in the initial stages of its dispersion is of great importance (at long distances all plumes eventually touch the ground).

In the near field (e.g. about 1 km away from the building) the plume dispersion does not follow a simple gaussian behavior, nor is the plume rise easily predictable. The dispersing plume interacts with various features of the wind flow around the building, such as the recirculation region and the aerodynamic disturbances from the building generally encourage downwash of the plume in the building wake [6], [8], [9]. Also, the plume does not generally arise from a point source, as in simple plume rise theory, but may be released from multiple sources.

In the cases of importance here, flows are dominated by buoyancy. Turbulence affects the rate of diffusion of mass, momentum and heat. The interaction of buoyancy and turbulence mixing in the rising plume is accounted for by an appropriately modified k-ε turbulence model.

A typical warehouse of interest in this work is of height 8-10m, of width 30-40m and of length 50-70m. Since we are concerned with pollutant sources located at the building itself, the flow field and the dispersion pattern around the building are extremely important. Although buildings of such a shape are common, available information on dispersion of pollutants is limited only to square shapes or rectangular shapes of width-to-height (w/h) aspect ratio up to about three [5]. Previous experimental and theoretical investigations have shown that the value of w/h and of the building inclination relative to the wind are the two main aspects of the building shape that affect flow and dispersion patterns ([2], [3], [4]).

Flows past buildings usually get strongly separated. Despite their frequent appearance, strongly separated flows are not extensively studied due to the difficulties encountered in their experimental and theoretical analyses. The basic characteristic of a strongly separated flow is an integral parameter that measures the length, L_R, of the recirculation region behind the building. L_R depends on the ratio of the thickness of the boundary layer in the approaching flow to the building height as well as on the geometry of the building itself [5], [6].

2. Theoretical model and computational approach

2.1 Governing equations and boundary conditions

To predict air pollution by fire products, the full (three dimensional) Navier-Stokes equations are solved together with the equations of energy and of turbulent flow properties and with the equations of conservation of chemical species. The Boussinesq approximation of the momentum equation is used to account for buoyancy effects, the density being taken as a function of temperature only in the gravity terms. The equations are shown in detail in [7] and [11].

At the solid boundaries of the computational domain, the no-slip condition is imposed; at the asymptotic outflow boundaries, the flow is taken as parabolic and at the top boundary as shearless. At open boundaries the pressure is taken as uniform.

To approximate reality better, two key ingredients were incorporated in our model to account for typical atmospheric conditions:

(a) The inlet velocity profile follows a 1/7-th power law, a good approximation of adiabatic atmosphere and relatively smooth surroundings [11].

(b) The inlet turbulence kinetic energy, k, is approximated according to Klebanoff's distribution of turbulent velocity fluctuations, that yields values of turbulence viscosity close to those measured under adiabatic atmospheric conditions [4].

2.2 Computational approach
The general purpose CFD code PHOENICS, along with the solution procedure called SIMPLEST, was used for the computational analysis.

The calculations were carried out by using a non uniform cartesian grid. The grid was fairly coarse near the outflow and inflow boundaries of the computational domain and fine enough near the prism walls and the mounting surface. Grid non uniformity and refinement of the discretization makes possible the resolution of the important flow features within the computational domain (such as around the prism).

The source of fire is modeled as a volumetric source releasing heat at a given, constant rate. The buoyancy forces are treated as impressed body forces and appear in the source term of the momentum equation. The velocity distribution close to solid boundaries is approximated in terms of the so-called "wall functions"[1].

The output of the model is the spatial distribution of the three velocity components, of the temperature and of the concentrations of fire products.

3. The Case under consideration

The computational case of concern is a building with a single hole at the centre of its roof which is amenable to thorough parametric studies, due to its relative simplicity. We studied the effect of buoyancy number, and of momentum number on flow and concentration distributions and, more importantly, on lift-off onset and ground clearing.

Computational domain:	1034m x 310m x 400m
Boundary layer height:	400m
Free stream velocity:	4 m/s
Model Building:	30m x 70m x 8m
Building height:	8m
Velocity at building height:	2.29 m/s
Computational grid:	84x29x31 (75,516) cells -- half of the real domain was discretized due to the symmetry of the problem.
Roof openings:	1 (at the centre of the roof)
Source :	mass 6 kg/s, enthalpy Q=3.06 MW, velocity 1m/s
Buoyancy Flux Number, F/U^3L:	0.0, 0.025, 0.05, 0.075, 0.1, 0.2, 0.3, 0.4
Momentum Flux Number, M/U^2L^2:	0.0, 0.05, 0.1, 0.2, 0.3, 0.5

The flow is considered steady because the ratio of width to height (l/h) is 7.85 (it is reminded that if the ratio is between 3 and 7, the flow is unsteady accompanied by vortex shedding).

Lift-off occurs at values F/U^3L close to 0.1. Onset of lift-off is deduced by the reduction of the concentration to approximately 6% of its value in the non-buoyant case (measured at the same point in the flow domain). The slope of the F/U^3L-versus-concentration curve changes markedly at lift-off (Figure 1). At $F/U^3L=0.3$ the concentration is 5.4‰ of its value in the non-buoyant case, which signals ground clearing. These values of the buoyant number at lift-off and clearing are in good agreement with values reported in literature and the measurements reported by Warren Spring Laboratory [9] [10]; namely, WSL reports lift-off onset at $F/U^3L=0.1$ and ground clearing at $F/U^3L=0.3$.

Increasing the momentum flux number reduces the concentration at ground level and along the centreline; however the order of magnitude of concentration values remains unchanged (Figure 2). The recirculation length increases with momentum flux number.

The recirculation length, L_R, increases from 2.8H in the non-buoyant case and remains constant at 3.7H at all values of F/U^3L. This could be attributed mainly to the gradual domination of the flow by buoyancy; when buoyancy becomes fully dominant, L_R reaches a constant value.

The effect of buoyancy on flow and concentration portraits and especially, on the recirculation flow past the building is clearly shown in Figures 3 and 4; buoyancy moves the plume upwards, away from the building wake.

4. Concluding Remarks

A computational fluid dynamics model is developed to be used for obtaining theoretical predictions of flow and dispersion of pollutants from warehouse fires.

A thorough parametric study of the effect of F/U^3L on flow and concentration is very useful because it reveals the influence of wind speed on plume lift-off and, consequently, on ground clearing. Lift-off was found to occur at $F/U^3L=0.1$, in agreement with values reported in the literature.

Momentum flux number affects concentration, but to an extent not as significant as that observed in buoyant flows.

Finally, the parametric analysis makes clear the significance of the effect of the inlet boundary conditions (vertical distribution of velocity, turbulence strength, boundary layer thickness) on flow and concentration fields.

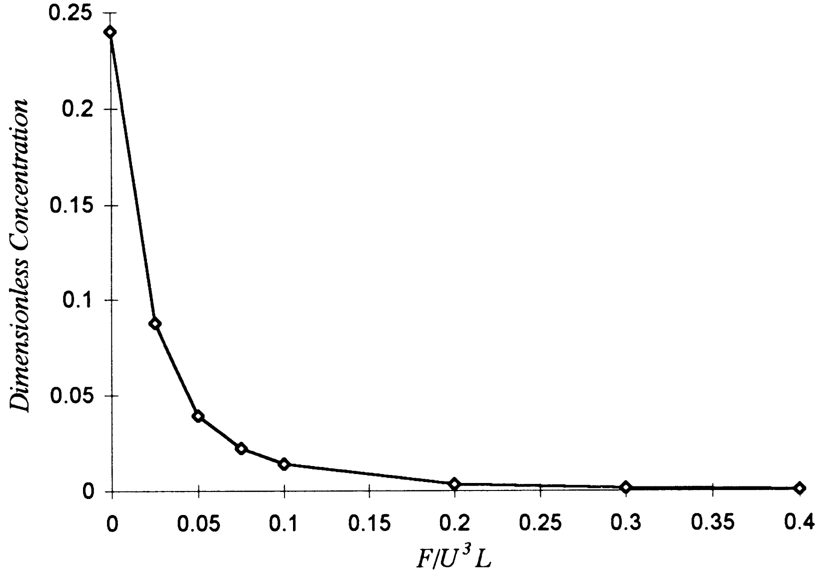

Figure 1: Effect of buoyancy flux number on ground level concentration

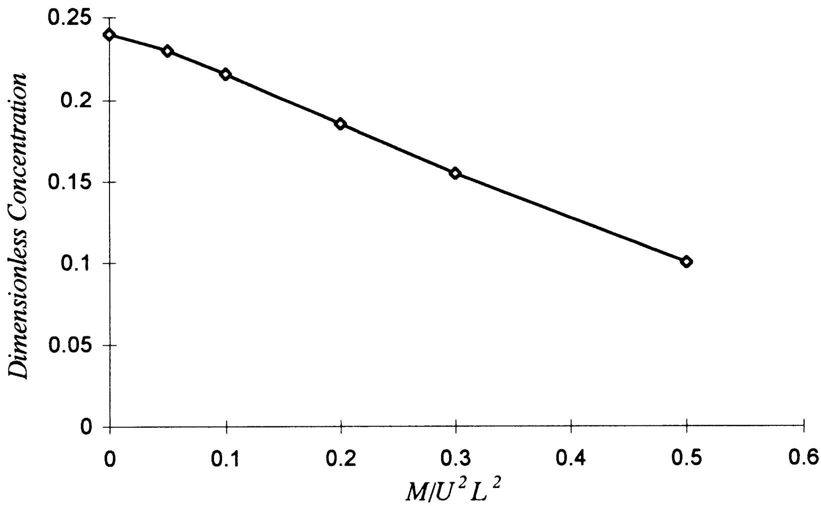

Figure 2: Effect of momentum flux number on ground level concentration

Transactions on Ecology and the Environment vol 6, © 1995 WIT Press, www.witpress.com, ISSN 1743-3541

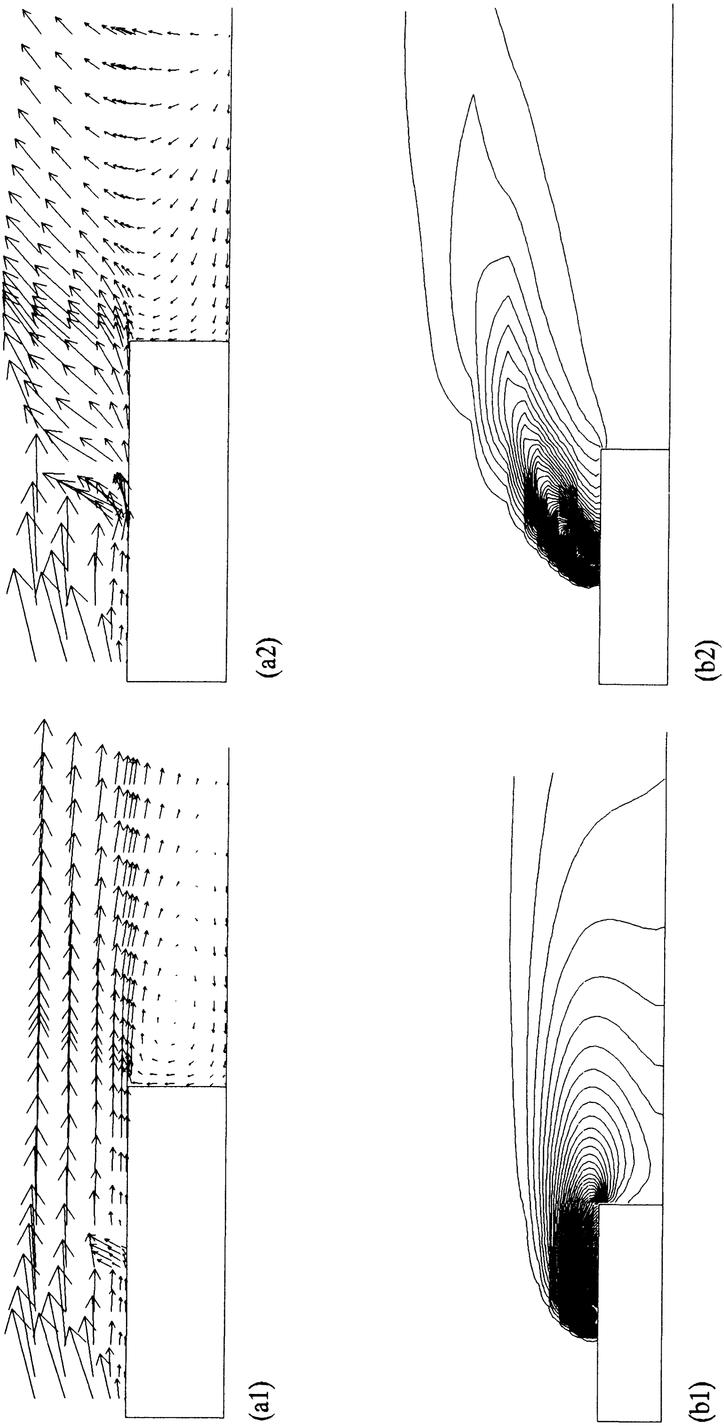

Figure 3: Flow and concentration fields. (a1) Non - buoyant flow field; (a2) Buoyant flow field (b1) Non - buoyant concentration field; (b2) Buoyant concentration field.

Transactions on Ecology and the Environment vol 6, © 1995 WIT Press, www.witpress.com, ISSN 1743-3541

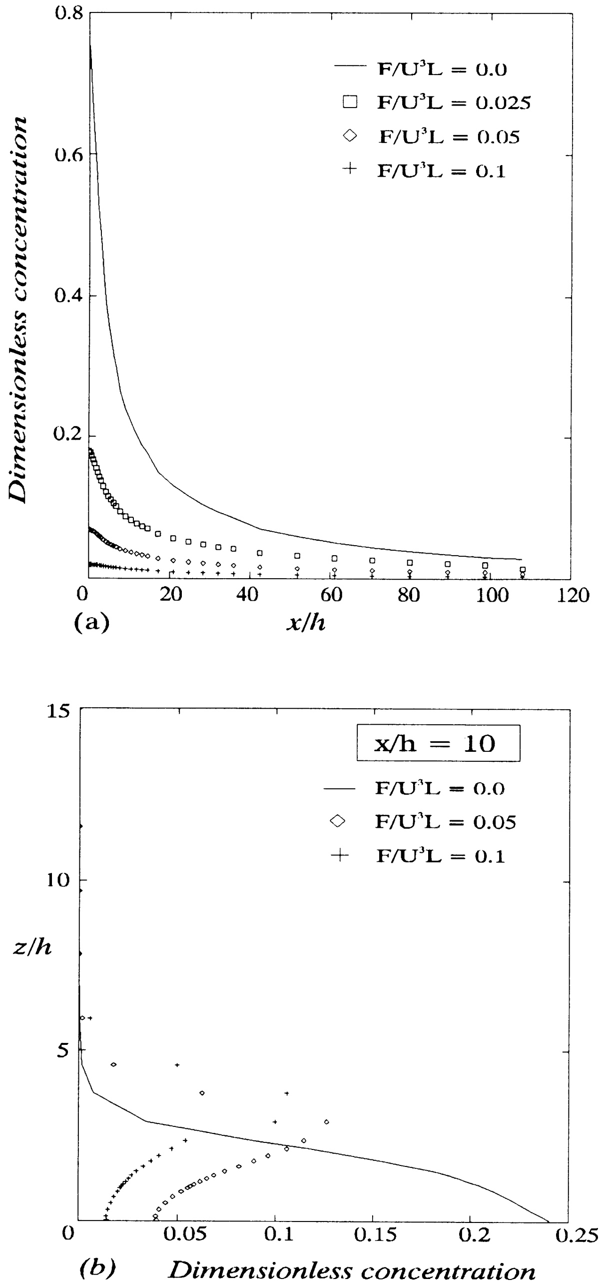

Figure 4: Effect of buoyancy on concentration
(a) Ground level concentration versus horizontal distance from the warehouse
(b) Concentration variation in the vertical direction

Therefore serious consideration should be given to the choice of the appropriate boundary conditions.

Acknowledgment

This work was partially supported financially by European Union Program STEP CT 90 0096.

References

1. Launder, B.E., Spalding, D.B., *Mathematical models of turbulence*, Academic Press, 1972.
2. Antoniou, J., Bergeles, G., Development of the reattachment flow behind surface mounted two dimensional prisms, *J. Fluids Eng.*, 1988, **110**, 127-133.
3. Bergeles, G., Athanassiadis, N., The flow past a surface mounted obstacle, *J. Fluids Eng.*, 1983, **105**, 461-463.
4. Trifonopoulos, D.A., Bergeles, G. C., Stable stratification effects on the flow past surface obstructions: a numerical study, *Int. J. Heat and Mass Fluid Flow*, 1992, **13**, No 2.
5. Castro, I.P., Robins, A.G. The flow around a surface mounted cube in uniform and turbulent streams, *J. Fluid Mech.*, 1977, **79**, 307-335.
6. Robins, A.G., Castro, I.P. A wind tunnel investigation of plume dispersion in the vicinity of a surface mounted cube. The flow field, *Atmospheric Environment*, 1977, **11**, 291-297.
7. Markatos, N.C., Malin, M.R., Cox, G., Mathematical modelling of buoyancy-induced smoke flow in enclosures, *Int. J. Heat Mass Transfer*, 1982, **25**, No 1, 63-75.
8. Xue, E., Hixara, T., Saito, Turbulence model of fire-induced air flow in a ventilated tunnel, *Int. J. Heat Mass Transfer,* 1993, **36**.
9. Hall, D.J., Kukadia, V., Marsland, G.W., Dispersion of plumes from warehouse fires, *Symposium of Institution of Chemical Engineers CIMAH Cases for Warehouses*, Chester, 1993.
10. Hall, D.J., Kukadia, V., Plume dispersion from chemical warehouse fires, *Second EC Seminar on Industrial Fires*, Cadarach, France 1994.
11. Miles, S., Cox, G., Christolis, M., Christidou, C., Boudouvis, A., Markatos, N.C., Modelling the environmental consequences of fires in warehouses, *4th International Symposium on Fire Safety Science*, Ottawa, Canada, 1994.