CFD Simulations of Dispersion Around Obstacles of Different Shapes

S. Andronopoulos[1], I. Mavroidis[2], A. Venetsanos[1] and J.G. Bartzis[1,3]

[1]National Centre for Scientific Research Demokritos, Institute of Nuclear Technology and Radiation Protection, Environmental Research Laboratory, Aghia Paraskevi, Greece

[2]Hellenic Open University, Patras, Greece

[3]University of Western Macedonia, Department of Mechanical Engineering, Kozani, Greece

Introduction

the study of flow and dispersion around isolated simple structures is very useful for detecting the fundamental characteristics of building influenced dispersion and for investigating routine or accidental releases of airborne hazardous or radioactive substances in *security-related applications*.

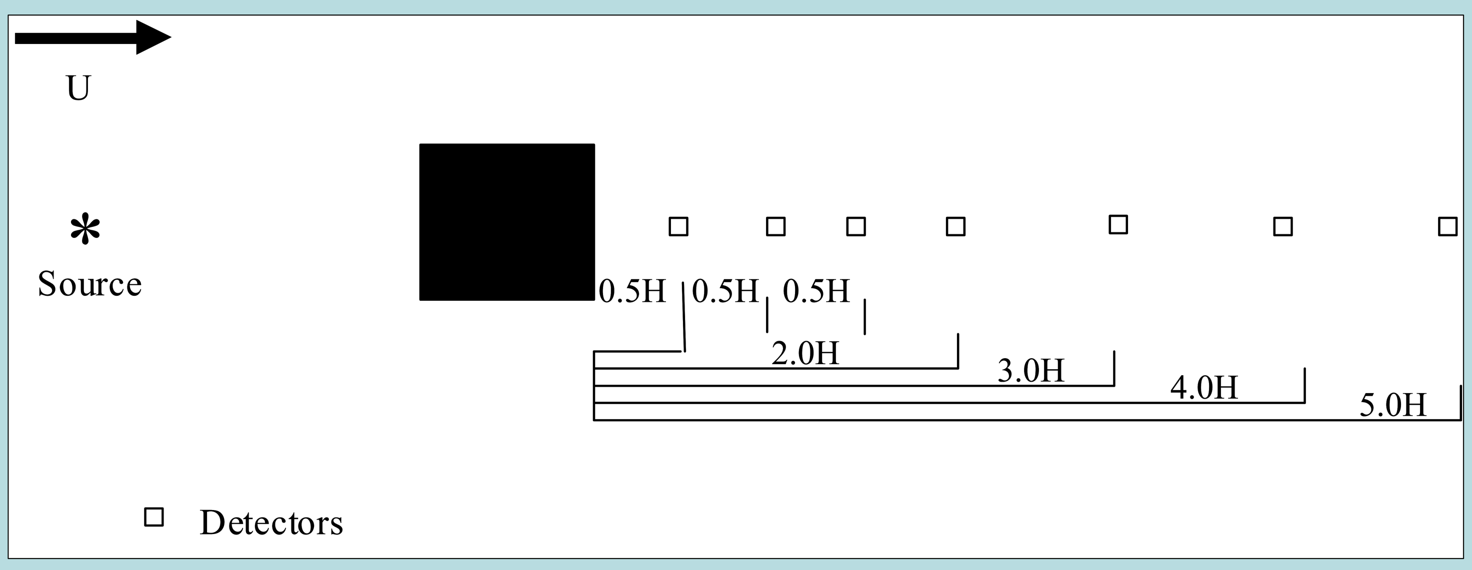
The present work aims at calculating mean concentrations and concentration fluctuations downwind of single obstacles for different source locations, in order to compare it with detailed results from wind tunnel experiments and to evaluate the performance of the CFD ADREA-HF model in the prediction of mean concentrations as well as concentration fluctuations in obstacle-obstructed flows.

Model Description

ADREA-HF is a finite volumes code that solves the Reynolds-averaged equations for the *mixture mass*, *momentum*, *energy*, *pollutant mass fraction* and the *variance of the pollutant mass fraction*. Turbulence closure is obtained through the eddy viscosity concept, which, in the simulations presented in this paper, is calculated by the *standard k-ε model*. The turbulent kinetic energy k and the dissipation rate ε are calculated by transport equations. For the pollutant concentration variance, a three-dimensional transport equation is also solved.

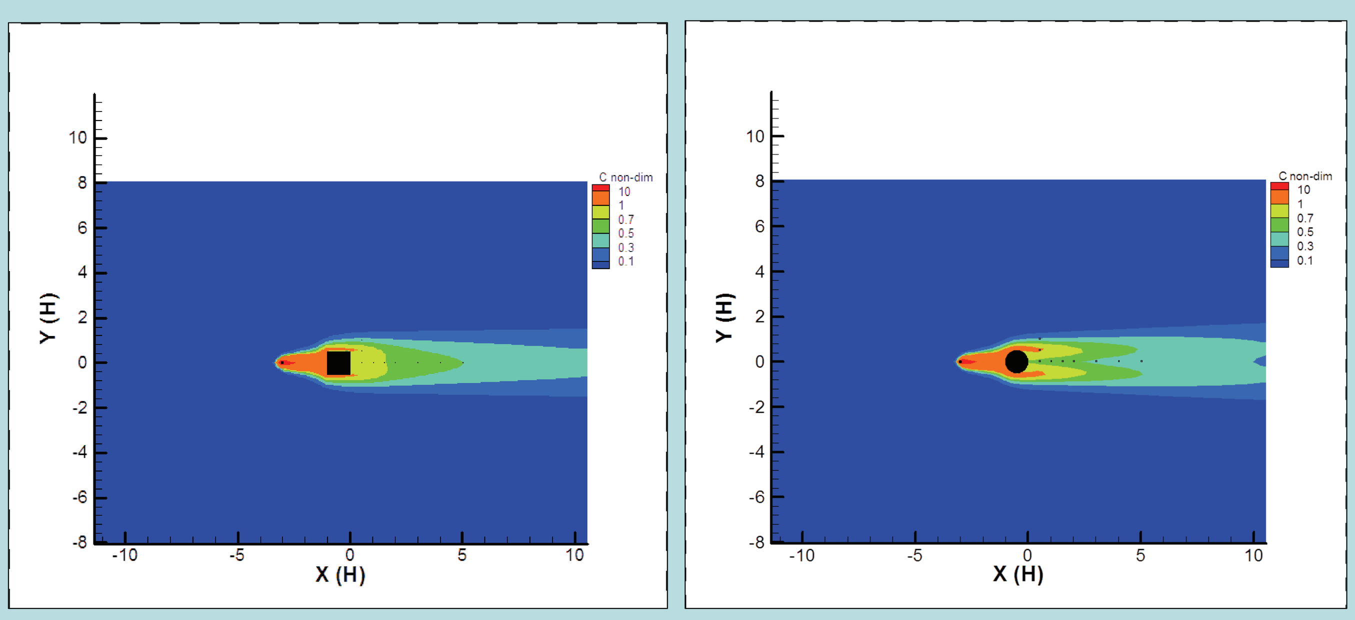
Experimental Data Base

- Wind tunnel experiments of tracer dispersion around single obstacles at the Building Research Establishment, at Cardington.
- Obstacles examined in this study: a cube of height H=0.15m and a right cylinder of height H and diameter D=0.15m
- Source locations investigated here : (a) 2 obstacle heights upwind, on the centreline and at half the building height (b) at the centre of upwind face, (c) at the centre of the roof, and (d) at the centre of the downwind face of the obstacle.

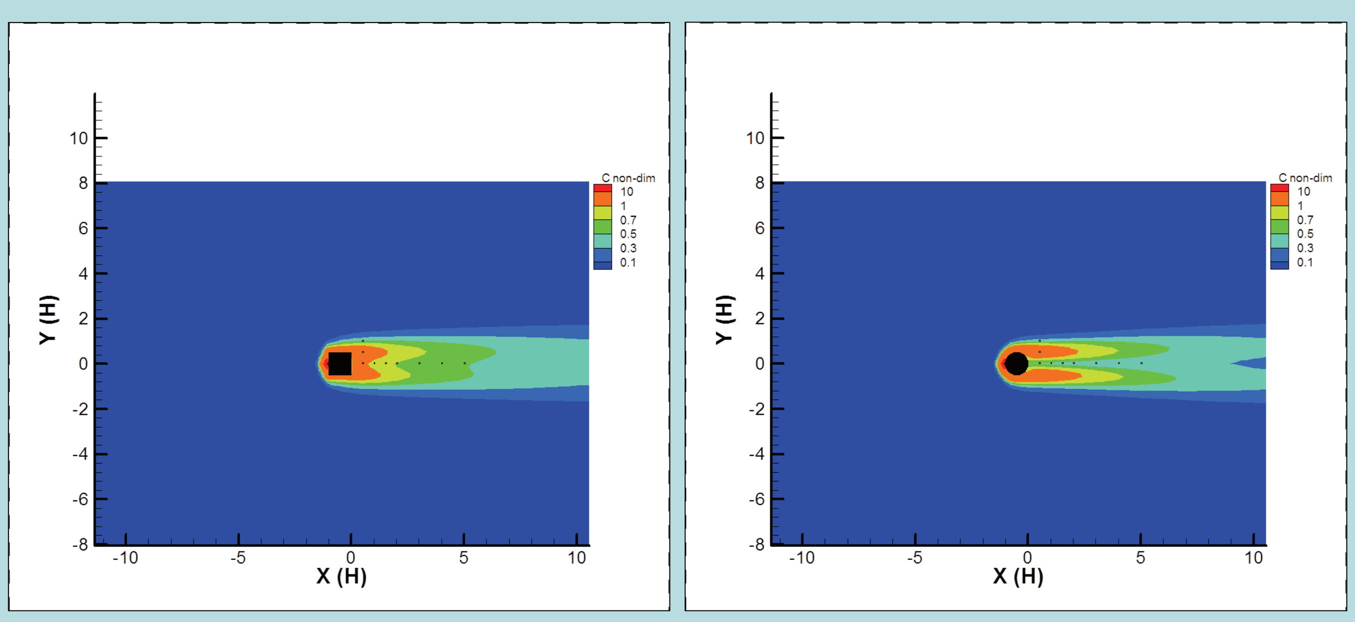


Plan view of the basic experimental configuration involving a single cube, with the source located 2.0H upwind

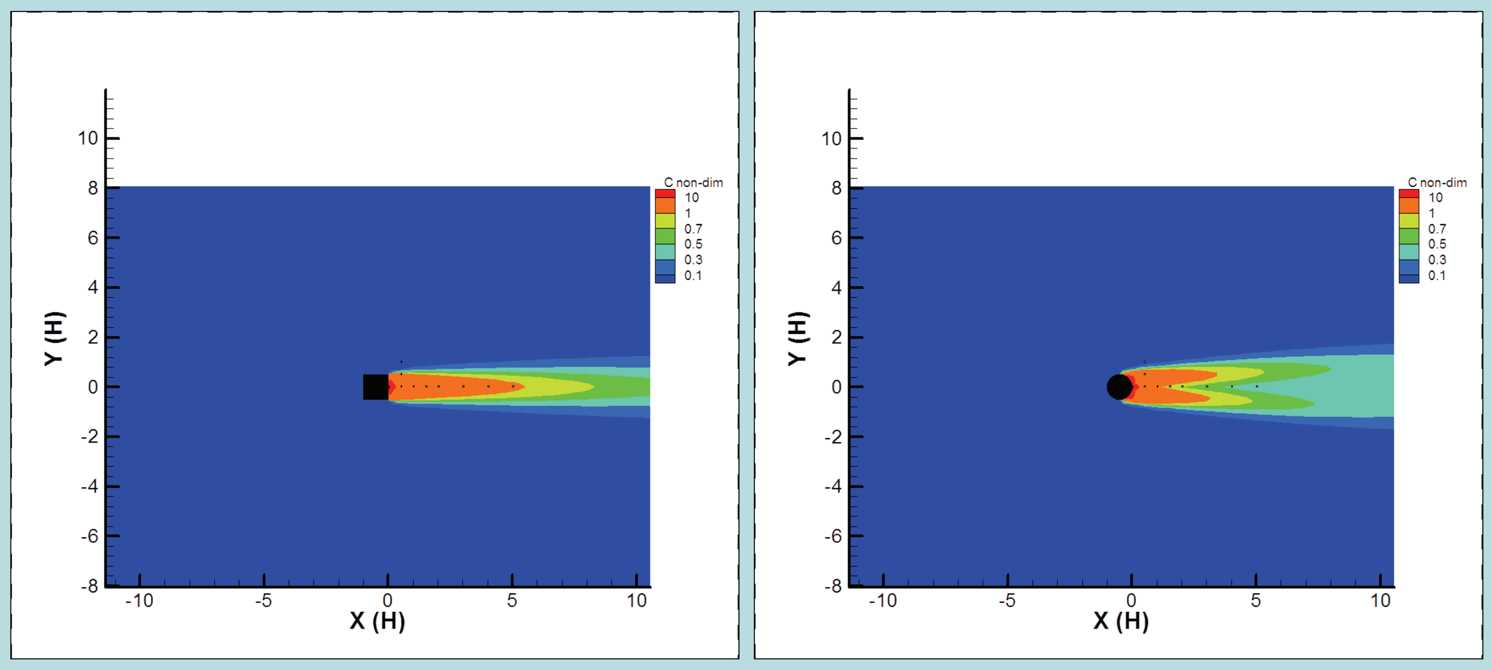
Results and Discussion



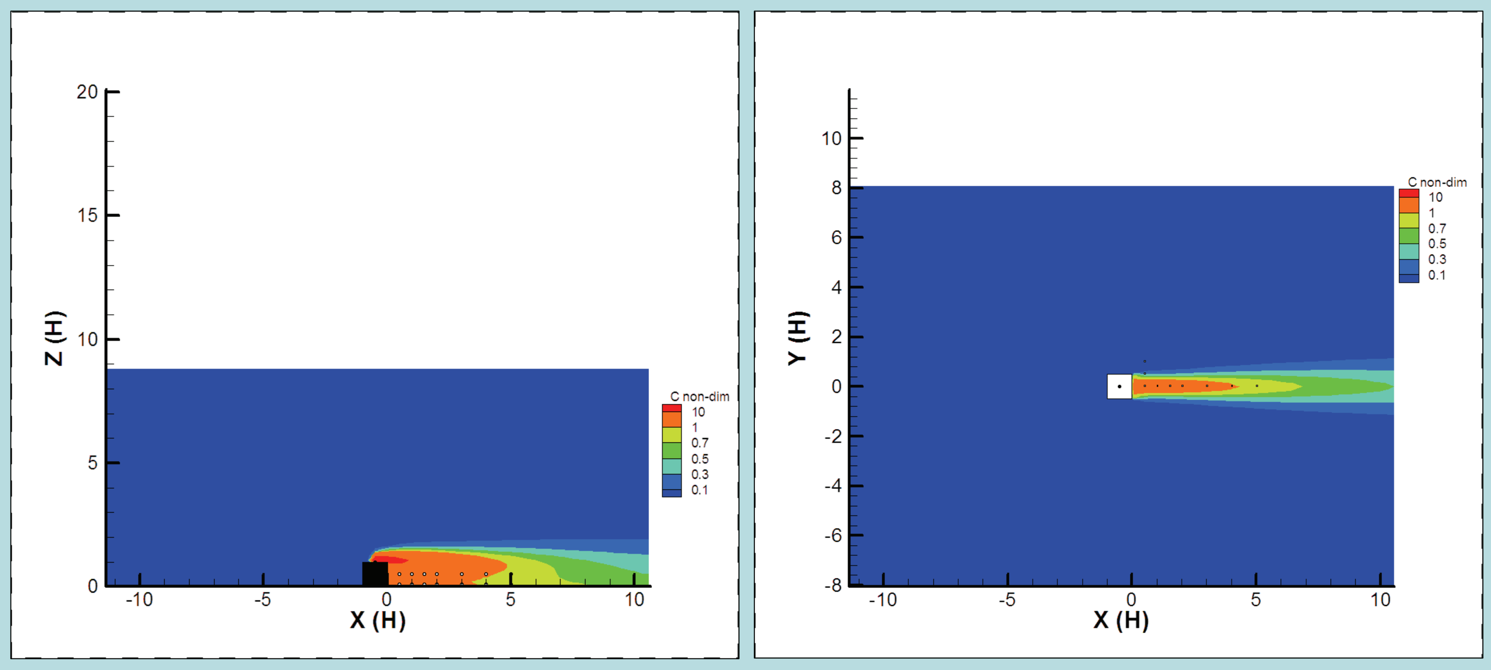
Contour plots of calculated non-dimensionalised concentration on the horizontal plane at the height of the gas source, located at a distance of 2H upwind of the obstacle: cube (left), cylinder (right); the bifurcation is more pronounced in the case with the right cylindrical obstacle



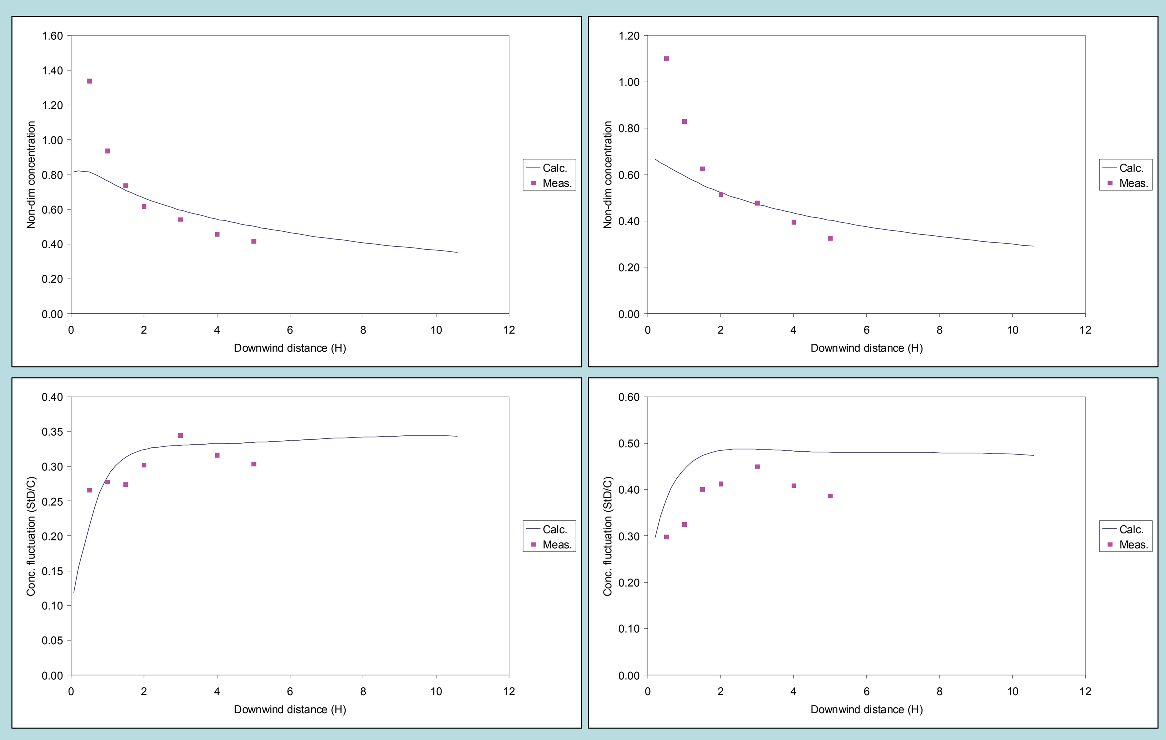
Contour plots of calculated non-dimensionalised concentration on the horizontal plane at the height of the gas source, located at the upwind face of the obstacle: cube (left), cylinder (right); again the bifurcation is more pronounced in the case with the right cylindrical obstacle



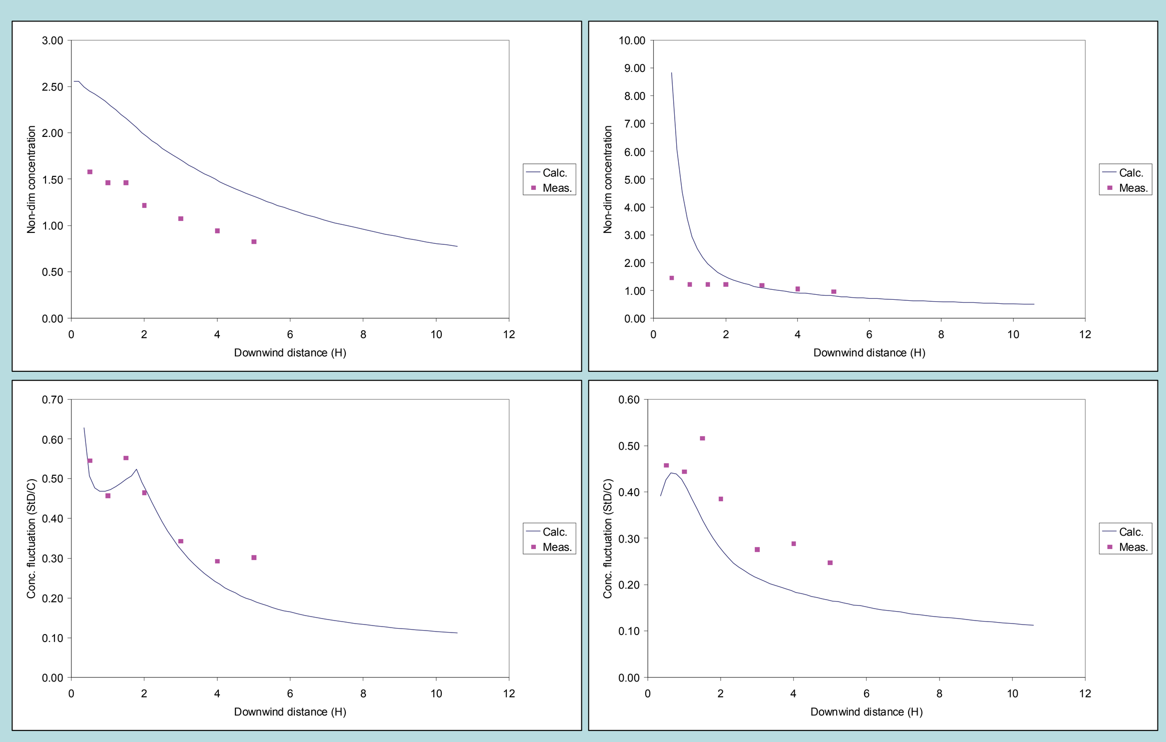
Contour plots of calculated non-dimensionalised concentration on the horizontal plane at the height of the gas source, located at the downwind face of the obstacle: cube (left), cylinder (right); no plume bifurcation is observed for the cube case



Contour plots of calculated non-dimensionalised concentration for the gas source located on top of the cubical obstacle: vertical plane (left), horizontal plane at the height of H/2 (right); the plume is mixed by the recirculation zone in the lee of the obstacle and touches the ground immediately downwind



Comparison of calculated and measured non-dimensionalized concentration (up) and concentration fluctuation intensity (down) in the along-wind direction, downwind of the obstacle, at height H/2, for source located 2H upwind: Cube, (left), cylinder, (right)



Comparison of calculated and measured non-dimensionalized concentration (up) and concentration fluctuation intensity (down) in the along-wind direction, downwind of the obstacle, at height H/2, for source located at the downwind side: Cube, (left), cylinder, (right)

Conclusions

Computational fluid dynamics (CFD) simulations of neutral gas dispersion wind tunnel experiments around isolated model buildings of cubical and cylindrical shapes are presented in this paper. Different gas source locations are examined. Patterns of computed concentration show a more pronounced plume bifurcation for the cases with the cylindrical obstacle. The model results for mean concentration and concentration fluctuation intensity are compared with measured data for model evaluation purposes, downwind of the obstacles at half the obstacle height and at ground level. The agreement is good for some cases, e.g., for the concentration at half the obstacle height and at distances 2 to 5 heights downwind, and poorer for other, as e.g., the concentration close to the obstacles immediately downwind. Fluctuations intensity is predicted rather well overall. Further research will focus on quantifying the effects of factors as turbulence modelling and boundary conditions on the level of agreement between model results and measurements in the wind tunnel.