H13-6
IMPACT OF FLOW OBSTACLES ON POLLUTANT DISPERSION UNDER CHANGING ELEVATED POINT SOURCE AND THERMAL STABILITY

Mohamed F. Yassin
Department of Environmental Technology Management, Kuwait University, P.O. Box 5969, Kuwait, Safat 13060, State of Kuwait

Abstract: The effect of flow obstacles on the pollutant dispersion with different elevated point source was investigated using computational fluid dynamics (CFD) models under different types of atmospheric stability within the atmospheric boundary layer. The results were compared with the experimental results obtained from the diffusion wind tunnel under different conditions of thermal stability. The flow and dispersion fields in the boundary layer in an urban environment were examined with different flow obstacles. The CFD models used for the simulation were based on the steady-state Reynolds-Average Navier-Stoke equations (RANS) with $k-\varepsilon$ turbulence models; standard $k-\varepsilon$ and RNG $k-\varepsilon$ models. The flow and dispersion data measured in the wind tunnel experiments were compared with the results of the CFD models in order to evaluate the prediction accuracy of the pollutant dispersion. The results of the CFD models showed good agreement with the results of the wind tunnel experiments. The results indicate that the turbulent velocity is reduced by the obstacles models. The maximum dispersion appears around the wake region of the obstacles.

Key words: Atmospheric turbulence; CFD models; Pollutant dispersion; Obstacles building model; Thermal stability

INTRODUCTION
The dispersion of potentially hazardous pollutants emitted from an elevated point source such as stack-gas is of great concern when addressing the possible consequence of such releases on the health and safety of people and the environment in the vicinity of the stack. Many variables affect the dispersion of emitted gases from stacks such as wind speed and direction, stability of the atmosphere, stack height, surrounding buildings, trees and topography, stack exhaust velocity and initial pollutant concentrations. There is no doubt that the ground-level concentration of a pollutant at near-source distance can be reduced by increasing the height at which the pollutant is released into the atmosphere. However, from a practical standpoint the benefits of lower pollutant concentration must be balanced against the increased cost incurred in construction of tall stacks.

A series of wind tunnel experiments and numerical simulations were performed in literature with the aim of simulating present conditions and understanding the phenomenon of air pollution diffusion emitted from the elevated point source. For instance, wind tunnel experiments were conducted for flow around a cube and prismatic obstacles. In such experiments, the variations of flow with obstacle dimensions were revealed and comparisons with those under a flat-plate condition, also including the stack gas diffusion were made (e.g. Wilson; 1979, Schumlan and Scire; 1991, Wilson and Lamb; 1994, Meroney, et al.; 1999). In an examination of the flow and concentration behind a model cube in the wind tunnel, Merony and Yang (1971) varied $V/U$, $H/H$ and wind direction. Recently, there have also been a number of wind tunnel studies of flow and/or dispersion around a single surface-mounted obstacles and small group of buildings in a turbulent boundary layer, notably the work of Snyder (1993) and Snyder and Lawson (1994). For instance, the numerical simulation of the flow and prediction of pollutant dispersion around obstacles buildings have been carried out by many authors using physical simulation such as in Halitsky (1963), Robins and Castro (1977), Wilson and Britter (1982), Wen-Whai and Meroney (1983), Huber (1989), Isaacson and Sandri (1990), Higson and Griffiths (1994), Saathoff et al. (1995), Macdonald et al. (1998), Mavroidis and Griffiths (2001), and Mfula et al. (2005).

Up to now, few studies have considered the impact of the flow obstacles on the pollutant dispersion under different types of thermal stability; stable, neutral and unstable conditions. The main goal of this work is to simulate the gaseous dispersion emitted around flow obstacles within an urban environment. In particular, the investigation is made into the effect of the different flow obstacles model building. Moreover, the proposed research aims to quantify numerically the benefits of the assessment concentration of gases in the leeward direction. The information provided by this research is thus expected to help in establishing guidelines to assist engineers to select the best design for the building configurations in urban environment.

2. EXPERIMENTAL SIMULATION
The experiment was performed in a closed thermal diffusion wind tunnel. For details of the wind tunnel and approaching flow, refer to Yassin et al. (2002). In the present study, a 1:500 scale model of the obstacles was used. Diffusion fields in the boundary layer were examined in three flow obstacle cases as follows: (1) Boundary layer without flow obstacles, (2) Boundary layer over two-dimensional plate obstacle and (3) Boundary layer over three-dimensional cubic building obstacle. The model stack, $H_s$ was located at $X/H=0$. Concentration predictions were made with the model stack’s height of $H_s/H=1$ and changed to $H_s/H=0.5$ for the same three cases, where, $H$ is the height of obstacle model. The schematic of flow obstacles and different elevated stack models are illustrated in Fig. 1.

The three different types of thermal stratification (stable, neutral and unstable) within the atmospheric boundary layer were created in the test section by controlling the inflow and wind tunnel floor temperatures. For details of the typical values for the various stability parameters, refer to Snyder (1981).

In the study of the concentration behavior, the general non-dimensional concentration $K$ was used as the ratio of the mean concentration $C$ at any point in the wind tunnel experiments to a reference concentration $C_o$:

$$K = \frac{C}{C_o} \tag{1}$$

$$C_o = \frac{q}{U_H H^2} \tag{2}$$

where, $H$ is the reference height, $U_H$ is the reference velocity, and $q$ is the contaminant release rate.
variables: namely, the turbulent kinetic energy, Launder and Spalding (1974). The conservation and momentum conservation. The conservation equation for species concentration of pollutants must also be solved together with the above-mentioned equations which describe the flow characteristics. A three-dimensional computational domain was used with the wind flow direction assumed to be perpendicular to the obstacles model. The simulated data were interpolated at the same grid points in the wind tunnel experiment. Two-dimensional plate and three-dimensional building obstacles model were used as the same in the wind tunnel experiment. A typical grid configuration in the near wake region of the obstacles model is shown in Fig. 3. The computational domain was 2 m long x 1.0 m wide x 1.0 m high, which was discretised as 90 x 70 x 56 grid. The model edge distance is 9H from the inlet domain, 15H from the outlet domain, 15H from the lateral domain, and 15H from the upper domain. Extensive tests of the grid intervals are carried out with increasing grid interval until further refinement is shown to be less significant. The grid intervals near the obstacles in the x-, y- and z-directions are Δx =0.16H, Δy = 0.16H, and Δz = 0.12H respectively. The grid chosen is finer close to obstacle and ground and then expands further away. The expansion ratio in the non-uniform grid is 1.1. A typical grid configuration of the obstacle model with different stack heights are shown in Fig 2. All calculations were performed using FLUENT 6.3.26, a commercial finite volume-based CFD models (FLUENT, 2006). On the other hand, the configuration was meshed using GAMBIT 2.4.6 software.

4. RESULTS AND DISCUSSION

4.1 Influence Obstacles Model of Flow Field

The CFD modelling and wind tunnel results in the flow patterns for all three atmospheric conditions; neutral, stable and unstable cases with three flow obstacle cases are shown in Figs. 3 to 8. The CFD simulation results were consistent reasonably well with the wind tunnel results obtained under thermal stability, except with cube model and without obstacle model under unstable condition. The discrepancy of the CFD simulation and wind tunnel results under unstable condition may be due to the fact that the floor of the wind tunnel is not heated after the obstacles. The buoyancy forces retard these profiles up to Z/H_{400} ≤ 2. Their effects are in opposition with the effects of the flow field by the turbulence. The mean velocity profiles in the leeward direction with the three flow obstacle cases in the stable and unstable boundary layer thickness are approximately the same, but the unstable profiles show increased velocity lower height due to increase in the turbulence, which augments momentum transfer from higher to lower levels. Furthermore, a thick internal boundary layer can be seen in the case with 2-D plate obstacle due to increased turbulence velocity in the three atmospheric conditions, while in the cases with 3-D cube obstacle and without obstacle, the internal boundary layer generated is thin and more or less the same in both cases. The reattachment length of the separated flows with 2-D plate obstacle is longer than that with the cubic model. The value of turbulence velocity in the leeward and vertical directions with the 2-D plate is higher than that without flow obstacles and with the 3-D cube.
All turbulent velocities are reduced by the stable stratification, the most important influence being observed on the mean velocities, which are higher for the unstable case than for the neutral and stable cases. The thickness of boundary layer with the three flow obstacles under stable and neutral conditions were approximately the same. While it was increased under unstable condition due to the increase of the turbulence velocity, which augments momentum transfer from higher to lower levels. Furthermore, the mean velocity with stable condition is smaller than that with unstable and neutral conditions.

4.2 Influence Obstacles Model of Dispersion Field

The dispersion concentration, $K$ were measured in the boundary layer at stack height, $H/H_s=1$ and 0.5 for all three atmospheric conditions with the three flow obstacle cases shown in Fig. 9. The CFD simulation predicted concentrations similar to these obtained with wind tunnel experimental results. The computed concentration diffusion with the standard $\kappa$-$\varepsilon$ model was quite in agreement with that obtained from the experimental results. The discrepancies in the spread concentrations between the wind tunnel and CFD models at some points in the vertical profiles may be due to low Reynolds number in the wind tunnel. The peak value of concentration for the three atmospheric conditions with the three flow obstacle cases ranging from 4 to 9 at a half stack height, where the effluent is emitted near the separation-reattachment region and created the downwashes due to the emission velocity from the stack being 10% of the free stream velocity. Dispersion concentration with the plate model is less than that with the cubic model due to the increased turbulence velocity with the plate model. While, at the half stack height, the concentration without flow obstacle is higher than that with the plate and cube model. The values of concentration with the plate model in the three atmospheric conditions are approximately the same at $H/H_s=0.5$ and 1; this is also found in the case with the cube model. However, the values of concentration without obstacle at $H/H_s=0.5$ are higher than that at $H/H_s=1$ because of the increased stream velocity. Therefore, any effluent rise due to buoyancy effects is to be considered separately in arriving at an effective source height. The maximum concentration is found around the wake region of the obstacles. Therefore, the spread concentration is high near the stack and getting smaller as it is distanced away from the stack. In general, the spread concentration for the unstable condition is higher than that of the neutral and stable atmospheric conditions (Ogawa, 1974) as shown in the present work, especially in the case of $H/H_s = 0.5$. This is due to the fact that the turbulent diffusion is decreased in unstable conditions.
5. CONCLUSIONS
The CFD models and wind tunnel experiment results described in this paper have provided significant data on pollutant diffusion emitted from different elevated point sources in an urban environment. The results obtained showed good agreement between the CFD simulation and wind tunnel results under thermal stability. Based on the results of the CFD models obtained in this study and the comparison with wind tunnel experiments, the following conclusions can be made: (1) The buoyancy forces have considerable effect on mean stream-wise and turbulence in the region up to \( \frac{Z}{H} \leq 2 \), (2) A thick internal boundary layer is generated in the case with plate obstacle, (3) The inner boundary layer is very thick around the wake region due to the turbulence mixing, (4) The peak concentrations for the three atmospheric stability conditions with the three flow obstacle cases are from 4 to 9 at the half stack height, (5) Dispersion concentrations in the unstable atmospheric case are higher than those in the neutral and stable cases, (6) Dispersion concentrations for the cubic obstacle are higher than that of the plate obstacle, (7) The value of pollutant concentration with height stack \( H_s/H =0.5 \) is higher than with \( H_s/H =1 \), (8) The maximum concentration is around the wake region of the obstacles

ACKNOWLEDGEMENTS
This work was carried out with funding from Kuwait University under project research No. WE02/08

REFERENCES

Meroney R.N. and Yang B.T. 1971: Wind tunnel Study on Gaseous Mixing Due to Various Stack Heights and Injection rates above an isolated structure, *CER71-72RM-BTY16*.


