

# **Computational Fluid Dynamics Simulation of Turbulent Flows and Pollutant Dispersion Around Groups of Buildings**

Neihad Al-Khalidy  
Vipac Engineers & Scientists  
Unit E1 -B Centre court, 25 Paul St. North, North Ryde, NSW 2113

The paper describes the computational fluid dynamic simulation of flow and pollutant dispersion around a number of buildings to the east and west of a proposed building. The pollutant sources are located at the roof of the proposed building. The design of the proposed building suggests locating 18 stacks and 2 vertical ducts through the roof of a number of selected areas. In this study we assumed a slightly heavier pollutant than air at the sources and the fluid flow was modelled by partial differential equations describing the conservation of mass, momentum and species concentration in three Cartesian coordinate directions for steady state conditions.

A geometrically 3-dimensional model of the proposed building with the stacks at its roof was assembled to capture the complex airflow pattern around the building and the pollution dispersion from the stacks. Also, incorporated into the analysis is a number of neighbouring existing buildings to the east and west of the proposed building. At the upwind free boundary inlet velocity profiles for the atmospheric boundary layer were derived from the Australian wind code.

The flow characteristics are seen to be captured well by the two-equation k- $\epsilon$  model. The pollutant concentrations were predicted at the chest level and at a range of elevations during near calm wind and windy conditions.

## **1. INTRODUCTION**

The flow and pollutant dispersion within the vicinity of buildings are extremely complicated in nature containing high turbulence intensities, recirculation, reattachment and dead zones where a pollutant can accumulate and help to create a potential chemical hazard. Little consideration has been given so far in publications to simulate pollutant distribution and wind patterns caused by complex building configurations. Recently, the use of numerical techniques including Computational Fluid Dynamics (CFD) to simulate pollutant distribution and wind patterns caused by building configurations has received much attention. CFD can be used as a tool to help the designer to examine the pollutant problem under various conditions.

Lee and Park (1994) estimated pollutant concentration in urban canyons using a two-dimensional flow model. Scanlon (1998) simulated the flow and pollutant dispersion around a cube. Baik and Kim (2002) investigated pollutant transport from urban street canyons using a two-dimensional flow and dispersion modelling.

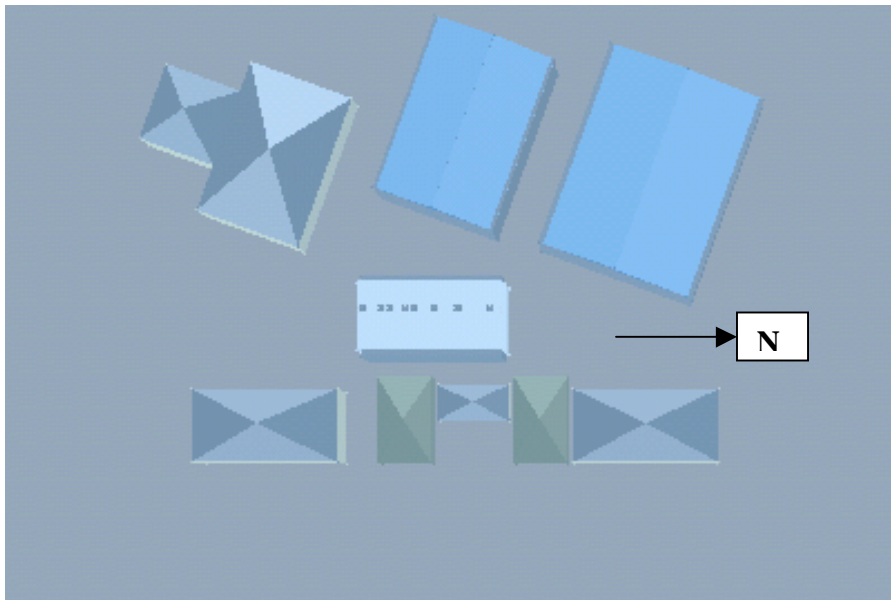
In this paper flow and pollutant dispersion around a number of buildings to the east and west of a proposed building are simulated during calm wind and windy conditions. The pollutant sources are located at the roof of the proposed building. The design of the proposed building

suggests locating 18 stacks and 2 vertical ducts through the roof of a number of selected areas.

## 2. MODEL DESCRIPTION

A geometrically 3-dimensional model of the proposed building with the stacks at its roof was assembled to capture the complex airflow pattern around the building and the pollution dispersion from the stacks. Also, incorporated into the analysis is a number of neighbouring existing buildings to the east and west of the proposed building as shown in Figure 1.

A calculation domain of 650 m length, 500 m wide and 80 m high was used for the CFD analysis. To minimise the solution time, the number of grids inside the calculation domain is reduced through grouping the fume cupboard exhausts. The design of the proposed building suggests locating 18 stacks with 3 m high and 0.31 m diameters through the roof of six selected areas. At each area the exhaust stacks are grouped into one exhaust stack with an equivalent diameter and fume mass flow rate.



*Figure 1: Geometric Model of the Proposed Building and Surrounding Existing Buildings*

## 2. 1 BOUNDARY CONDITIONS

The CFD study was undertaken to estimate the pollution concentrations during near calm and windy conditions:

- Wind Load Condition: An estimate of average wind velocity was determined for two prevailing wind conditions namely, southwest and northeast winds. At the upwind free boundary inlet velocity profiles were derived from the Australian Wind Code AS1170.2. Surrounding terrain was classified as Category 2 to the east and the west of the proposed building. A profile velocity inlet was provided 300 m to the west and east of the building and more than 250 m to the south and north of the building. For all cases the following boundary conditions were assumed representing the hourly mean wind speed with height with a one year return period:

- Height 0 to 20 m: Resultant velocity=5 m/s
- Height 20 to 40 m: Resultant velocity=5.8 m/s
- Height 40 to 100 m: Resultant velocity=6.8 m/s

At the downwind and upper free boundaries constant pressure boundary conditions were applied. The fume flow being released continuously from the stacks.

- Near Calm Air Condition: A magnitude of wind speed of 1 km/hr was chosen to represent the calm condition. In this study only the east condition is investigated. A pressure boundary condition of zero referenced to ambient static pressure is applied downstream to the west of the proposed building with a fume flow being released continuously from the stacks.

### 2.1.1 Pollution Emission Rate

The following pollution emission rates were included in the 3-dimensional analysis during windy and near calm wind conditions:

- A pollution concentration of unity is assumed at the pollutant sources (i.e. exhaust stacks and ducts). The contaminant at the source is modelled as one variable called CO.
- The pollutants were assumed to be slightly heavier than air at the sources. However, pollutants of a similar density than that of air are also investigated.

### 2.1.2 Other Boundary Conditions

The following additional boundary conditions were used

- Turbulence quantities (kinetic energy and dissipation rate) were calculated from empirical relationships
- A wall function data group was used to avoid using a very fine mesh near the wall and improve turbulent flow simulation

## 2.2 MATHEMATICAL REPRESENTATION OF THE PROBLEM

The CFD model solves the continuity, momentum and species concentration in three directions for steady (time invariant) and incompressible flow:

- Conservation of Mass

$$\frac{\partial}{\partial x_i} (\rho u_i) = 0 \quad (1)$$

- Conservation of Momentum

$$\frac{\partial}{\partial x_j} (\rho u_i u_j) = - \left( \frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \right) + \rho g_i + F_i \quad (2)$$

- Conservation of scalar species

$$\frac{\partial}{\partial x_j} (\rho u_i C) = \frac{\partial}{\partial x_i} \left( \frac{\nu_i}{S} \frac{\partial C}{\partial x_j} \right) \quad (3)$$

Where  $\rho$  is the density,  $u$  is the velocity,  $p$  is the static pressure,  $\rho g$  and  $F$  are the gravitational body and external body forces,  $\tau_{ij}$  is the stress tensor,  $C$  is the pollutant concentration and  $\nu$  is the kinematic viscosity and  $S$  is the volumetric heat source.

The Reynolds stresses in equations 2 are modelled using  $\kappa$ - $\epsilon$  model. The following transport equations are used to calculate  $\kappa$  and  $\epsilon$  terms:

$$\frac{\partial}{\partial x_i} (\rho \kappa u_i) = \frac{\partial}{\partial x_j} \left[ \left( \rho + \frac{m_t}{S_k} \right) \frac{\partial \kappa}{\partial x_j} \right] + G_k + G_b - \rho \epsilon - Y_M + S_k \quad (4)$$

$$\frac{\partial}{\partial x_i} (\rho \epsilon u_i) = \frac{\partial}{\partial x_j} \left[ \left( \rho + \frac{m_t}{S_e} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_{1e} \frac{\epsilon}{k} (G_h + C_{3e} G_b) - C_{2e} \rho \epsilon \frac{\epsilon^2}{k} + S_e \quad (5)$$

Where  $G_k$  is the generation of turbulent kinetic energy due to the mean velocity gradient,  $G_b$  is the generation of turbulent kinetic energy due to the buoyancy,  $Y_m$  is the contribution of the fluctuating dilation in compressible turbulence to the overall dissipation rate,  $\mu_t$  is the

turbulent viscosity,  $C1\varepsilon$ ,  $C2\varepsilon$  and  $C3\varepsilon$  are constant,  $\sigma_k$  and  $\sigma_\varepsilon$  are the turbulent Prandtl number,  $S_k$  and  $S_\varepsilon$  are user defined source term. Model constants are detailed in Launder B. and Spalding D, 1972.

## 2.3 DISCRETISATION

The software package utilised in the current CFD analysis is the commercially available code Phoenix. The CFD model solves continuity, energy and momentum equations described in section 2.2 in the computational domain to predict the steady state airflow and pollutant dispersion at the proposed and neighbouring buildings.

For the current analysis more than 680 000 non-uniform grid cells covered the computational domain. The following techniques were used for discretisation

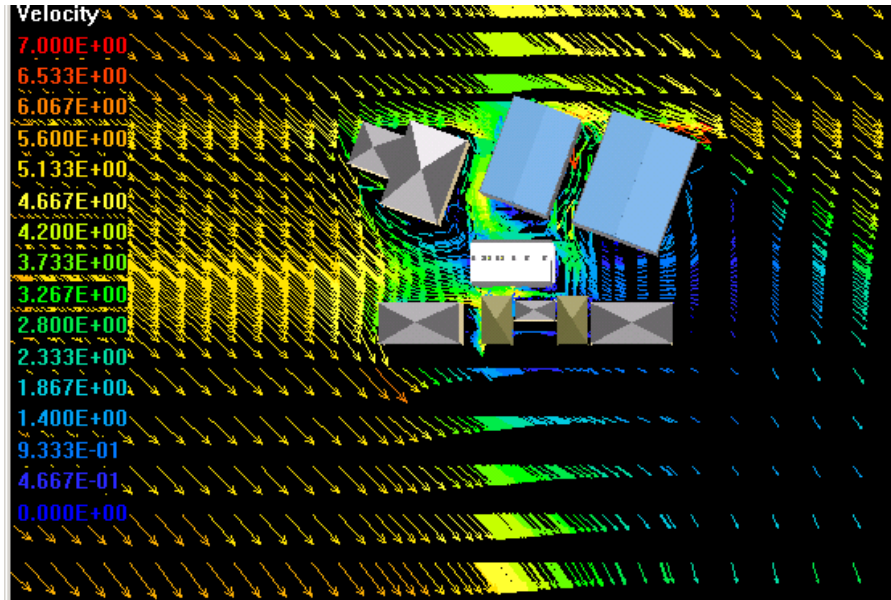
- Hybrid upwind differencing numerical scheme was used for discretisation of convective transport
- An iterative procedure was used to estimate the air velocity into three directions, pressure profile, turbulence parameters and pollutant concentrations. For the pressure velocity coupling Phoenix employs a global solver based on the SIMPLE algorithm (Patankar, 1989)
- Relaxation parameters were specified to stabilise the solution process. For this model we found that the convergence can not be obtained without using relaxation technique to control the variable rate of change between iterations

## 3. RESULTS AND DISCUSSIONS

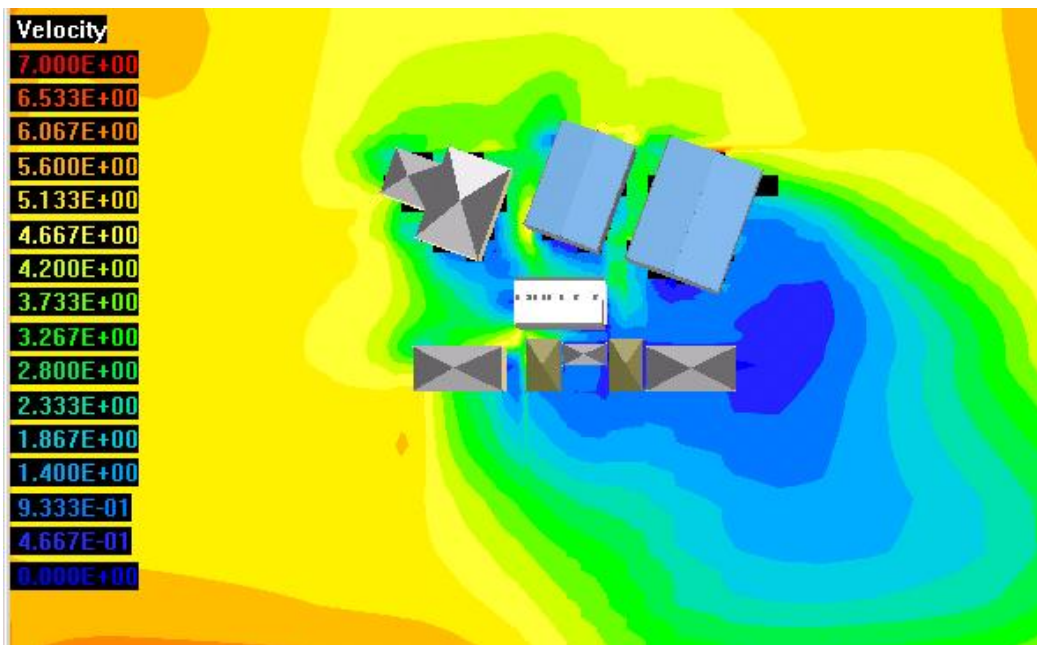
Steady state simulations of airflows and pollutant concentrations at neighbouring buildings and the building have been carried out using the geometric model described above. The results in Figure 2 to Figure 5 are presented for south-westerly winds where the average velocity at 1.5 m is 5 m/s. In this study inlet velocity profiles were derived from the Australian Wind Code AS1170.2.

Figure 2 (velocity vector) and Figure 3 (Velocity Contour) show a plan view of the flow field 1.5 m above the ground level (typical chest level). Flow characteristic is seen to be captured well by the CFD model and the following can be concluded:

- The flow around the proposed and neighbouring buildings generates an expected boundary layer profile
- The flow is accelerated near the corners as expected
- A stagnation region is located immediately downstream of the proposed building and a number of neighbouring buildings. This is related to wind direction and angle of attack relative to the local building orientation



*Figure 2: Velocity Vector in Horizontal Section at 1.5 m Elevation – Southwesterly Winds*



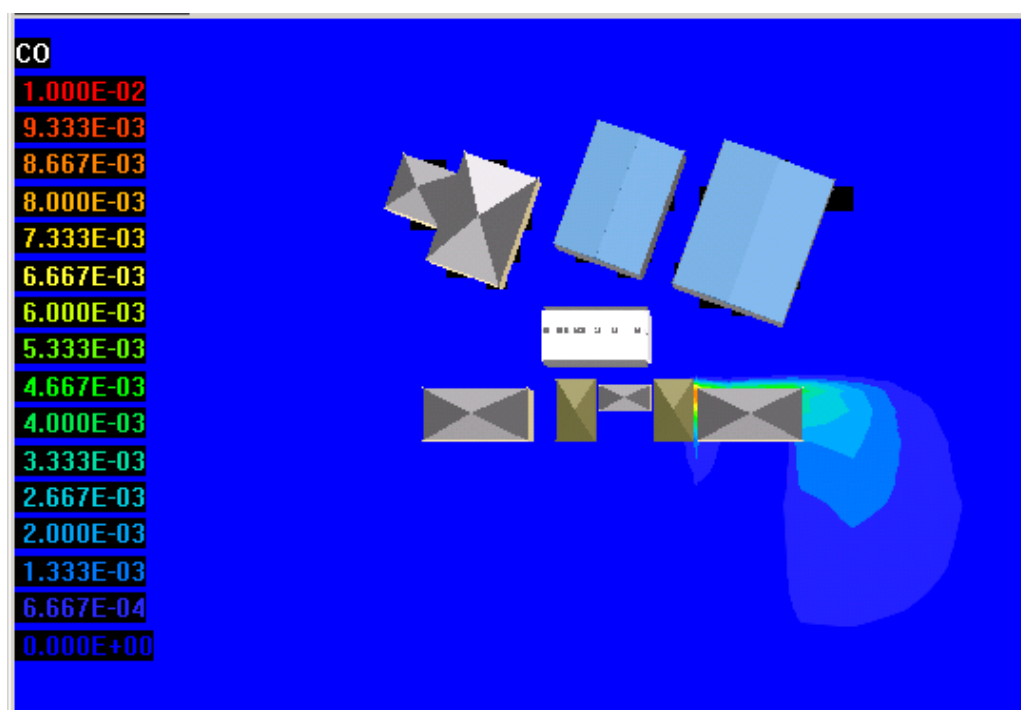
*Figure 3: Velocity Contour in Horizontal Section at 1.5 m Elevation – Southwesterly Winds*

Figure 4 shows the concentration profiles in a horizontal section at the chest level. One can see that the pollutants are affected by the downwash flow and also transported away from the proposed building. The concentration profiles are plotted on a scale between 0 to 1%.

Pollutants with 1% source concentration are seen to accumulate near the dead zones created by buildings to the east of the proposed building. The horizontal section of concentration profiles at a height of 15 m is shown in Figure 5. It can be seen that the pollutant dispersed to a wider region and the concentration is increased to 1.7%. A fume iso-surface is also provided in Figure 6 to indicate the approximate spread of pollutants in three-dimensions. The red contour shown represents the pollutants with 1% source concentration.

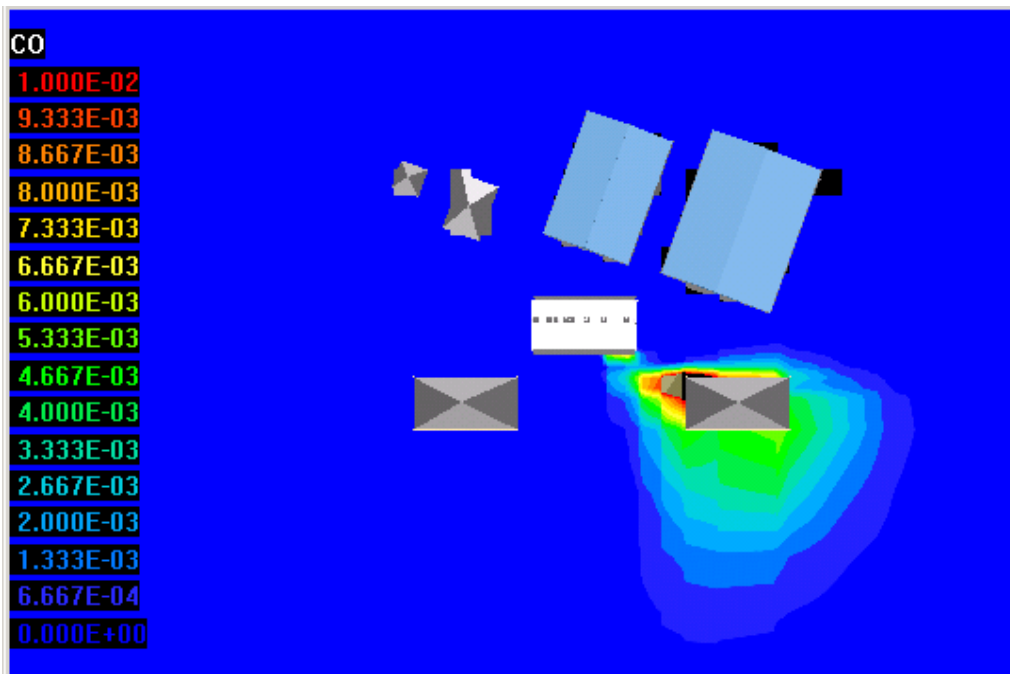
When we rerun the above case using pollutants of a similar density than that of air. We found that the concentration is reduced to about 0.3% at the chest level as shown in Figure 6 in comparison to 1% when a slightly heavier pollutant was used (Figure 7). In general, the concentration profiles are similar in both cases with heavier pollutants have more concentrations near the dead zones.

The actual pollutant concentration (Xylene in this study) around the buildings is calculated from multiplying the pollutant concentration ratios by the emission concentration. The emission concentration of Xylene at human height level should be less than 80 ppm<sup>1</sup>. In a study where human volunteers were exposed to 100 and 200 ppm for 3-7 hours no effect or blood pressure were observed (Ogata et al, 1970).

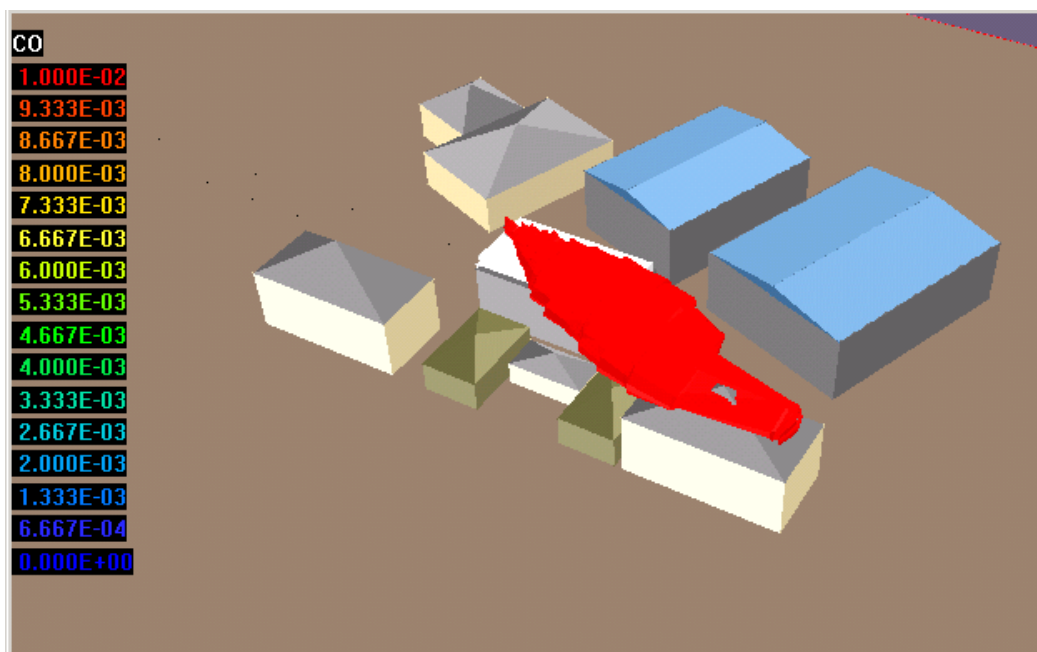


*Figure 4 Concentration Profile in Horizontal Section at 1.5 m Elevation*

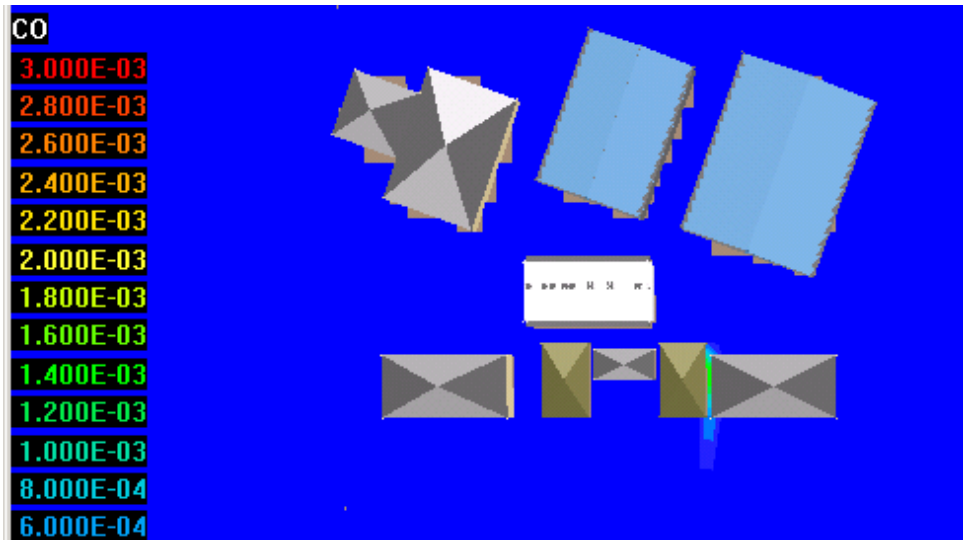
<sup>1</sup> <http://www.nohsc.gov.au>, Exposure Standards of Xylene , 22 Nov. 2002



*Figure 5: Concentration Profile in Horizontal Section at 15 m Elevation*

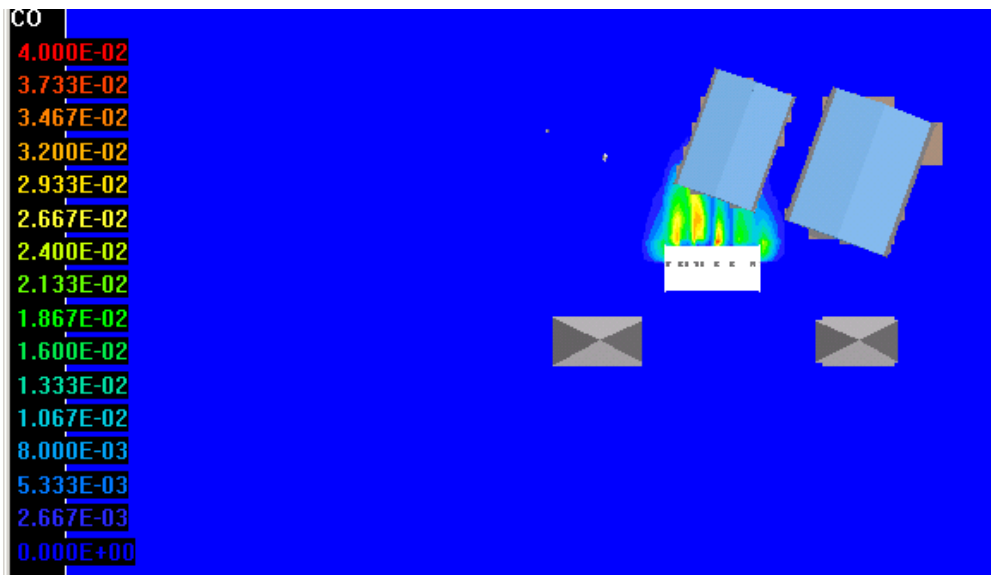


*Figure 6: Concentration Profile- Isosurface of 1% Source Concentration*



*Figure 7 Concentration Profile in Horizontal Section at 1.5 m Elevation – Pollutant of a Similar Density Than That of Air Was Used*

The fume concentration profile in a horizontal section during calm wind condition is shown in Figure 8. The pollution concentrations seen to change between 2 to 4% source concentration near the roof of the proposed building and a neighbouring building to the west of the proposed building.



*Figure 8: Concentration Profile in Horizontal Section Near the Roof of the Proposed Building*

#### 4. CONCLUSIONS

Flow fields and pollutant dispersal around a number of buildings to the east and west of a proposed building have been predicted using computational fluid dynamics analysis. The flow characteristics are seen to be captured well by the two equation k- $\epsilon$  model. The pollutant concentrations were predicted at the chest level and at a range of elevations during near calm wind and windy conditions. The CFD results will be validated against the measurement data when the proposed building is completed and operated.

#### 5. REFERENCES

1. Australian Building Code AS 1170.2, 1989
2. Baik J. and Kim J., On the Escape of Pollutant From Urban Street Canyons, Atmospheric Environmental, Vol. 36, pp. 527-536, 2002
3. CHAM- Polis from Phoenix V3.4, 1999
4. Launder B. and Spalding D., *Lectures in Mathematical Models of Turbulence*. Academic Press, London, England, 1972.
5. Lee Y. and Park H., Parameterisation of the Pollutant Transport in Urban Street Canyons, Atmospheric Environmental, Vol. 28, pp. 2343-2349, 1994
6. Patankar S., Numerical Heat Transfer and Fluid Flow, Hemisphere, New York, 1980
7. [http:// www.nohsc.gov.au](http://www.nohsc.gov.au), Exposure Standards of Xylene, 22 Nov. 2002
8. Ogata Met al, Urinary Excretion of Hippuric Acid and m- or p-Methylhippuric Acid in the Urine of Persons Exposed to Vapours of Toluene and m- or p-Xylene as a Test of Exposure, Br J. Ind Med, 27 pp. 43-50, 1970
9. Scanlon T., A numerical Analysis of Flow and Dispersion Around a Cube, University of Strathclyde Publication, Scotland, pp. 1-9, 1998