COMPUTATIONAL FLUID DYNAMICS
Simulation of Turbulent Flows and Pollutant Dispersion Around Groups of Buildings

presented by
Dr. Neihad Al-Khalidy
Specialist Consultant
Fluid and Thermal Technologies
Background

- The analysis of flow and dispersion of pollutants around buildings is important from an environmental point of view.
- Examine pollutant problem under different scenarios based on a detailed fluid flow Analysis.
Strategy

- CFD Model Construction
  - Wind data
  - Pollutant sources
- Flow Simulation
- Pollutant Dispersion Prediction
- Probability Analysis
- Recommend modifications to the duct heights, dimensions or velocities to achieve the recommended criteria
- Compliance with Occupational Health & Safety Commission limits at human height levels and living area
Geometry – Proposed Building
Geometry – Surrounding buildings
Geometry – Surrounding buildings
Boundary Condition - Wind Data

- Still wind condition
- Windy condition
- At the upwind free boundary inlet velocity profiles were derived from the Australian Wind Code AS1170.2
Boundary Condition - pollutant Data

- The maximum pollution emission rate at each stack is 450 l/s
- The maximum mass flowrate of Xylene fume is 1200 l/s at each of the two ducts through the roof located near the plant room
- A pollution concentration of unity is assumed at the pollutant sources
- The pollutants were assumed to be slightly heavier than air at the sources
CFD Modelling

- Phoenics Software
- Navier-Stokes equations for continuity, momentum, energy and species concentration
- Steady-state
- Incompressible
CFD Modelling

- Standard K-ε model
- 680000 unstructured grid cells
- A Hybrid numerical approach to discretise the convective term in the governing equations.
- SIMPLE algorithm for the pressure – velocity coupling.
- Relaxation parameter to stabilize the solution processes
- convergence 20-32 hours CPU

Phoenics User Conference on CFD May 2004
CFD Flow Results

South-Westerly Wind Conditions

Phoenics User Conference on CFD May 2004
CFD Flow Results

South-Westerly Wind Conditions

Phoenics User Conference on CFD May 2004
CFD Pollutant Dispersion Results

- Concentration profile in a horizontal section at the chest level.
- Downwash effect
CFD Pollutant Dispersion Results

1. Concentration profiles in a horizontal section at the 15 m elevation

2. The pollutant dispersed to a wider region and the concentration is increased to 1.7% source concentration.
CFD Pollutant Dispersion Results

1. concentration profiles in a horizontal section Through the outlet of the stack
2. The pollution concentrations seen to increase to 3% source concentration near the roof of the building.
CFD Pollutant Dispersion Results

Concentration Profile - Iso-surface of 1% Source Concentration – Pollutant Slightly Heavier Than Air
CFD Pollutant Dispersion Results

Concentration Profile - Iso-surface of 1% Source Concentration – Pollutant of a Similar Density Than That of Air Was Used

Phoenics User Conference on CFD May 2004
CFD Flow Results

North-Easternly Wind Conditions

Phoenics User Conference on CFD May 2004
CFD Pollutant Dispersion Results

Concentration Profile in Horizontal Section at 1.5 m Elevation
CFD Pollutant Dispersion Results

Concentration Profile in Horizontal Section at 15 m Elevation
CFD Pollutant Dispersion Results

North-Easterly Wind Condition
CFD Pollutant Dispersion Results

Calm Wind Condition – near the roof of the building
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 $k-\varepsilon$ 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 analysis has offered a comprehensive range of output including velocity distribution, pressure profile and turbulence levels. Subsequent testing of the modified duct system has validated the approach using CFD analytical tools.
- The CFD results will be validated against the measurement data when the proposed building is completed and operated.