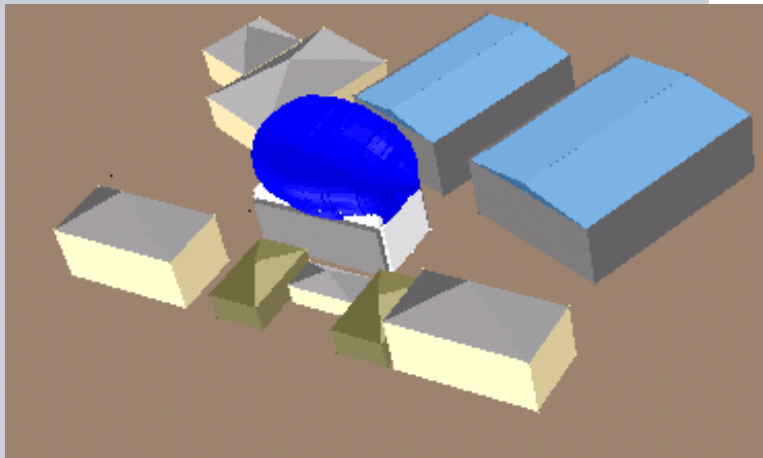


# Vipac Engineers & Scientists Ltd

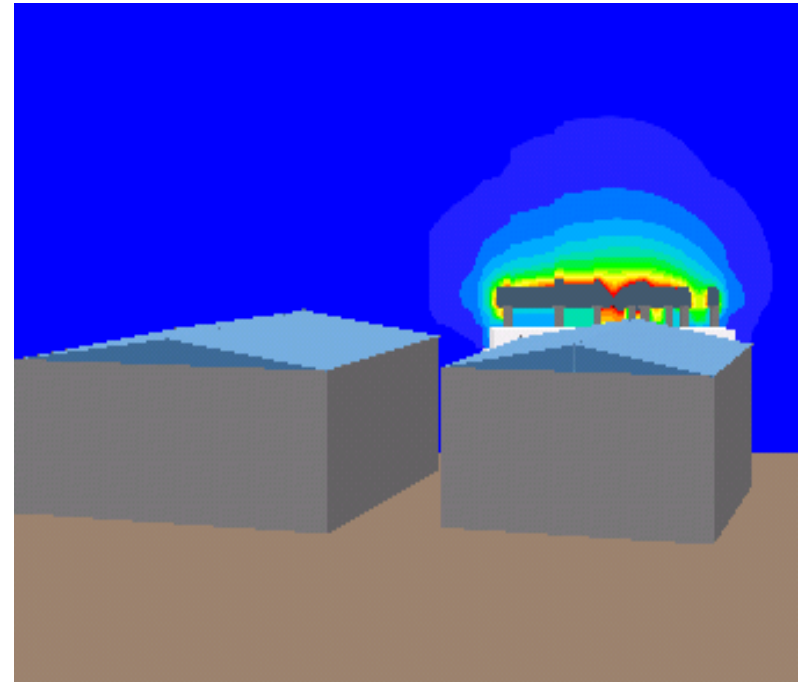
## **COMPUTATIONAL FLUID DYNAMICS Simulation of Turbulent Flows and Pollutant Dispersion Around Groups of Buildings**



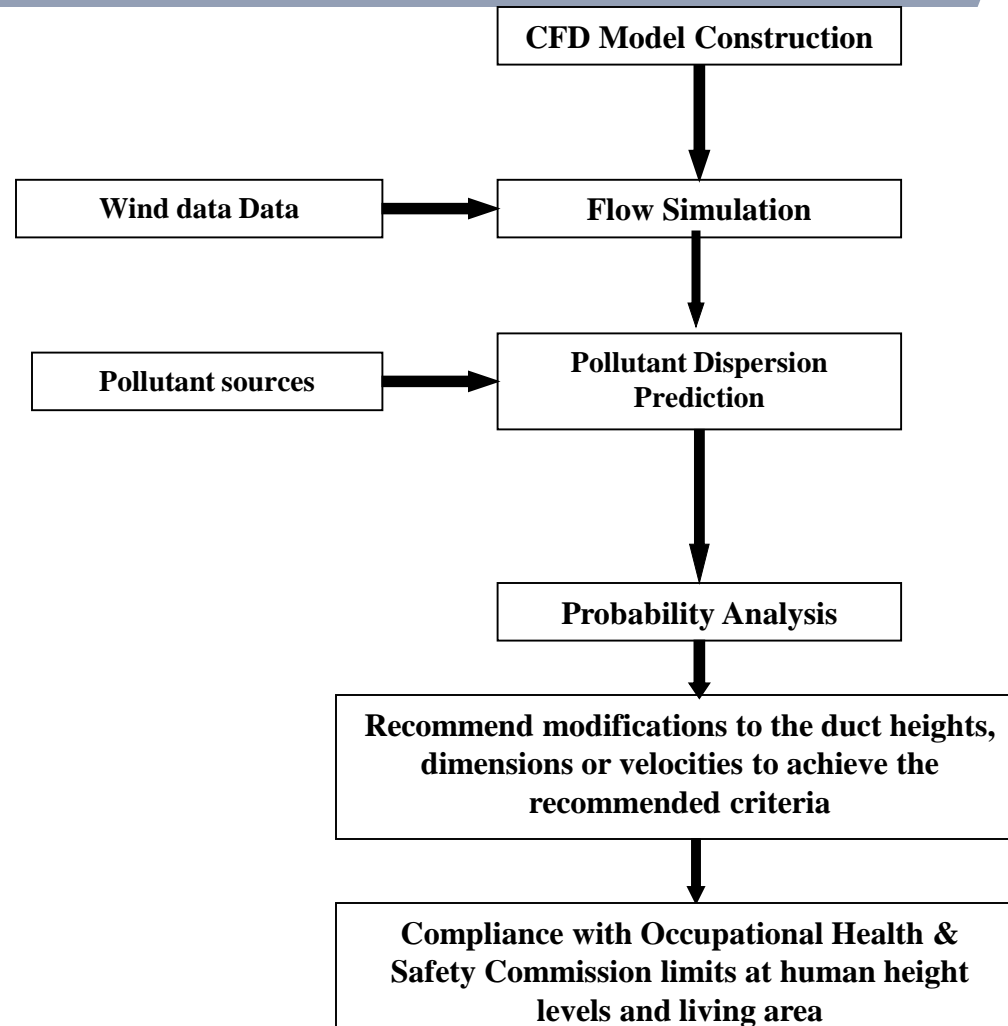
presented by  
**Dr. Neihad Al-Khalidy**  
Specialist Consultant  
Fluid and Thermal Technologies

# 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



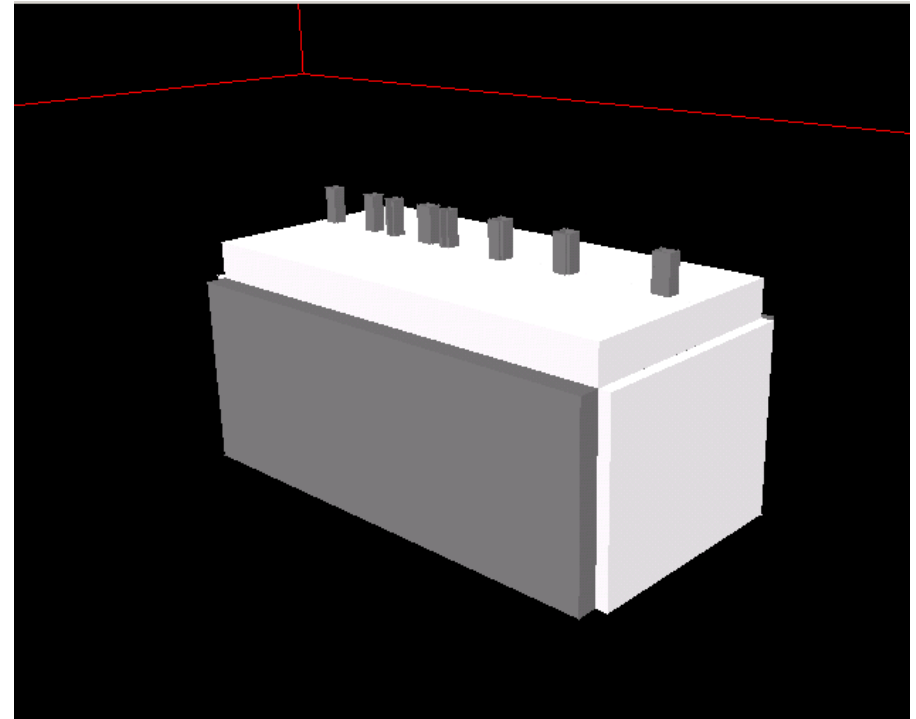
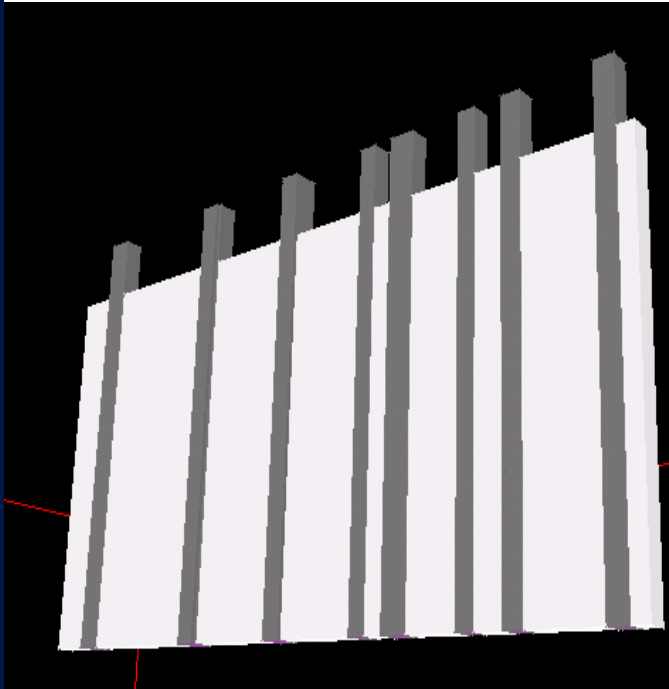
# Strategy



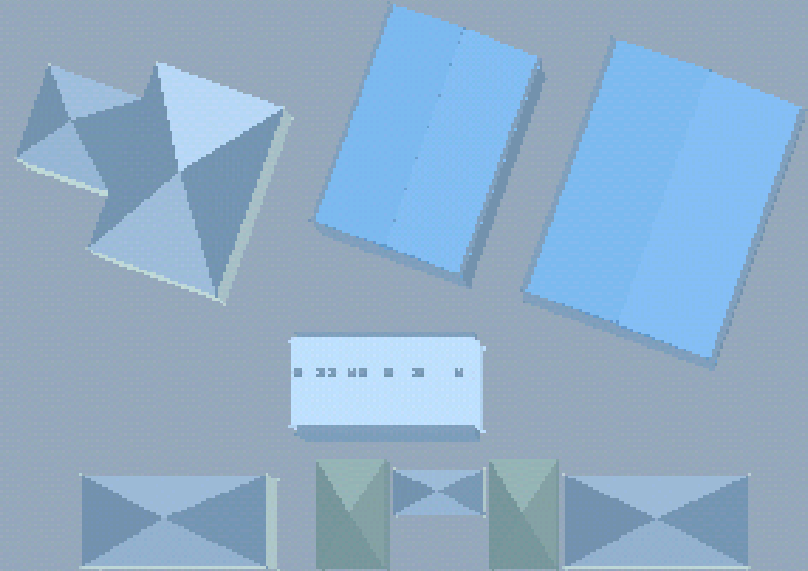
Phoenix User Conference on CFD May 2004



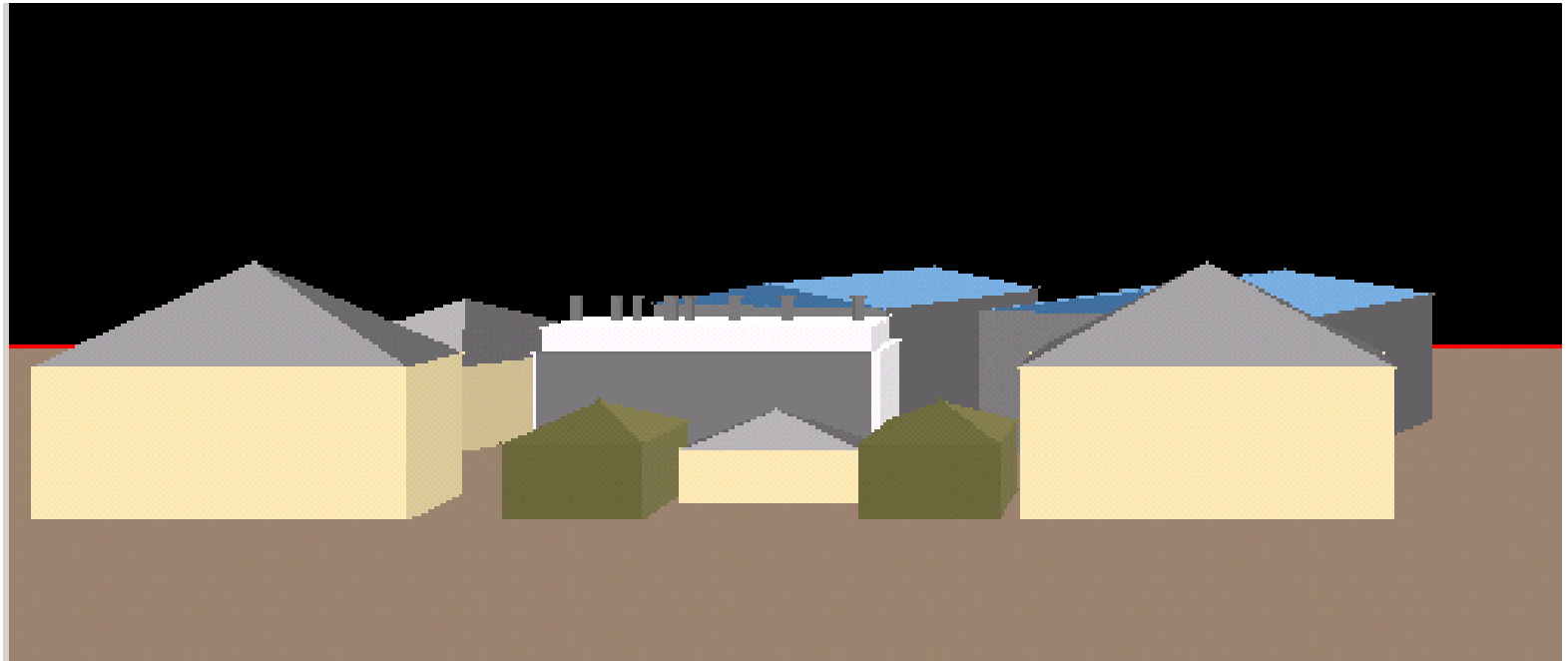
# Geometry – Proposed Building



# Geometry – Surrounding buildings

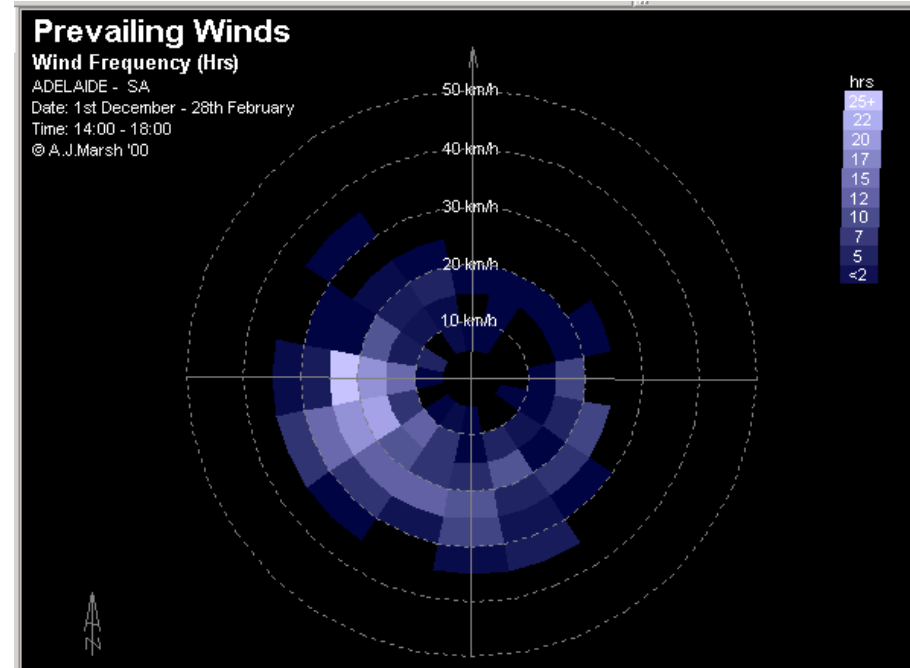


# Geometry – Surrounding buildings



# 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



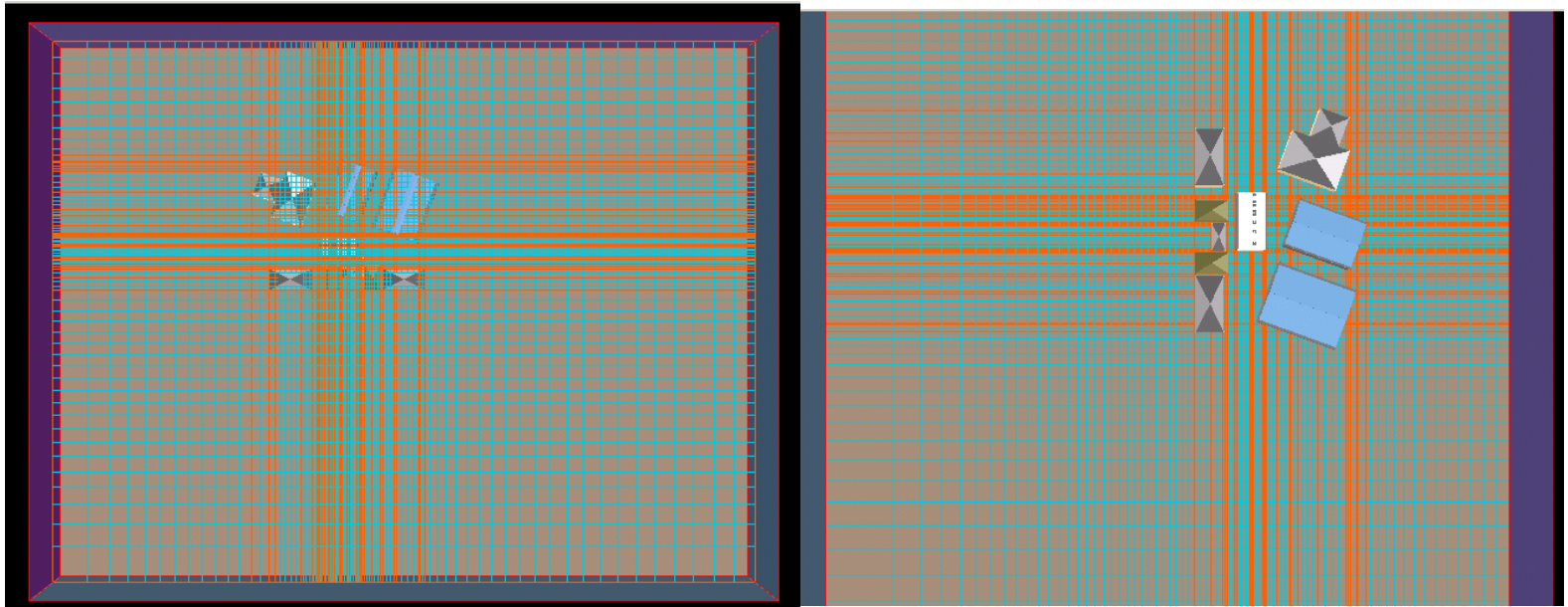
# CFD Modelling

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

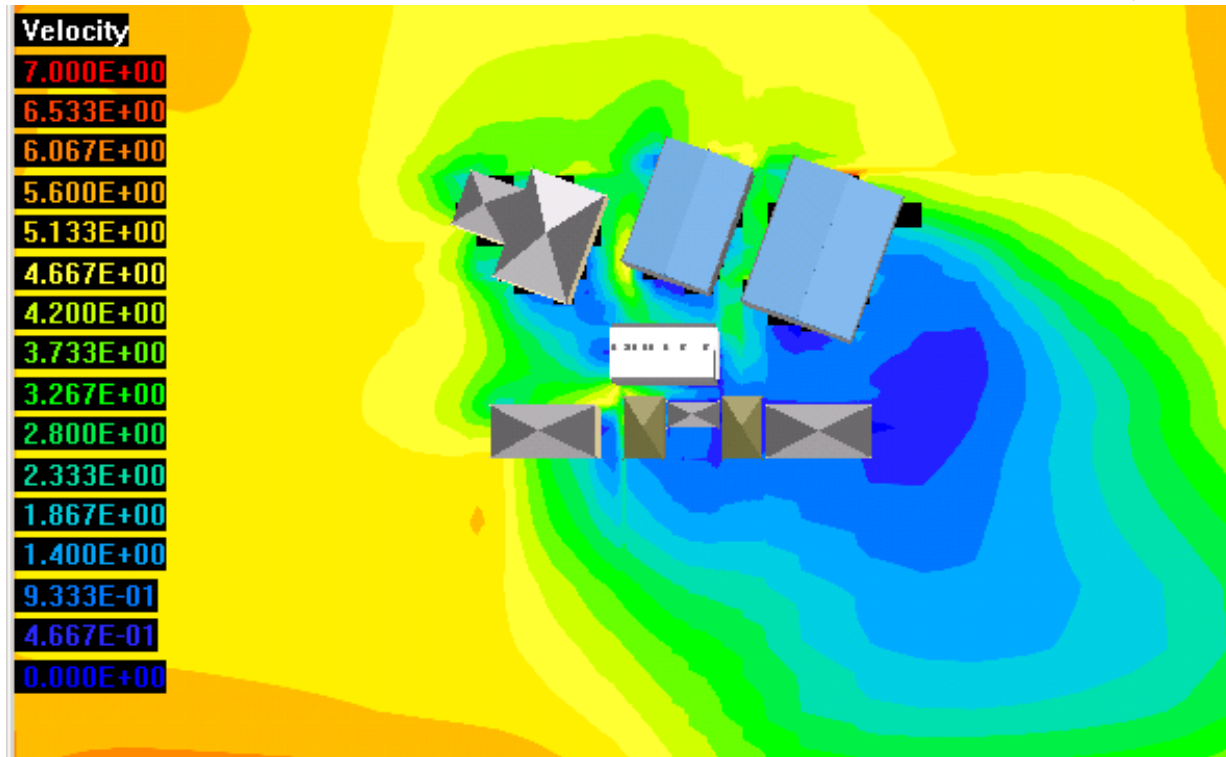
# CFD Modelling

- Standard K- $\epsilon$  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

# Sensitivity Analysis



# CFD Flow Results

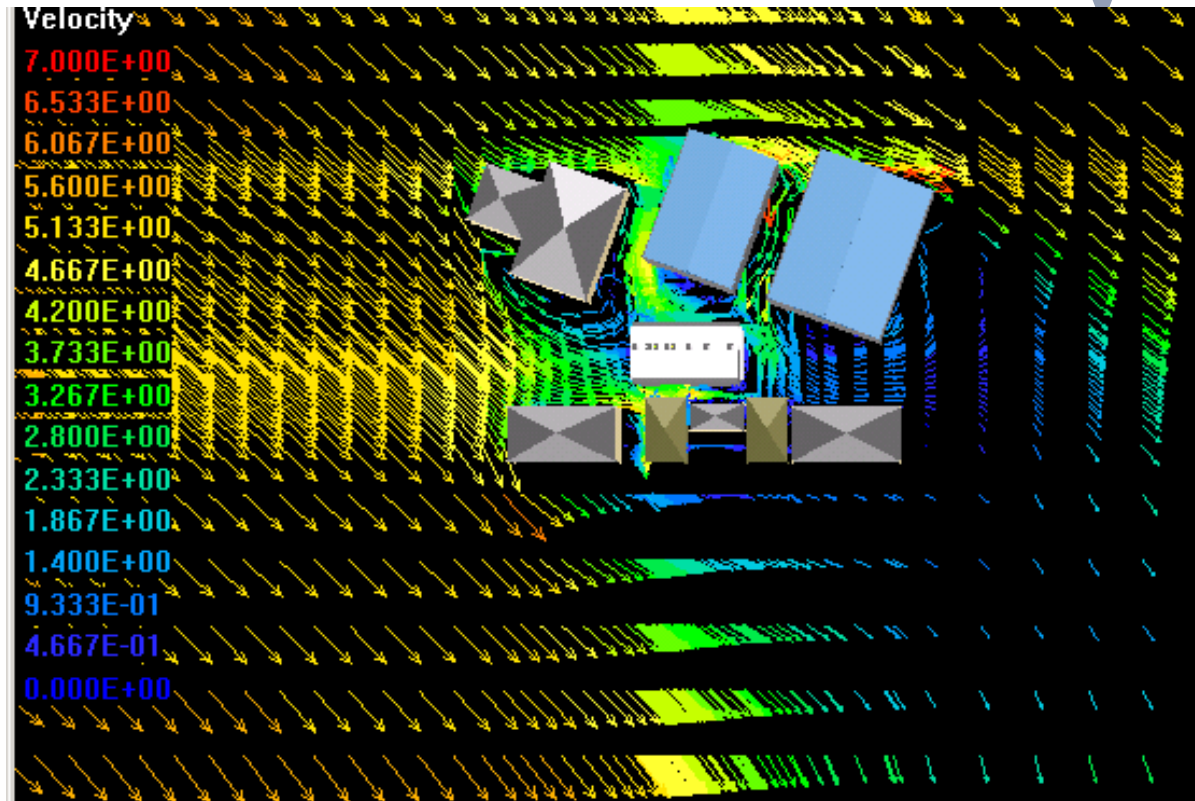


South-Westerly  
Wind Conditions

Phoenics User Conference on CFD May 2004



# CFD Flow Results



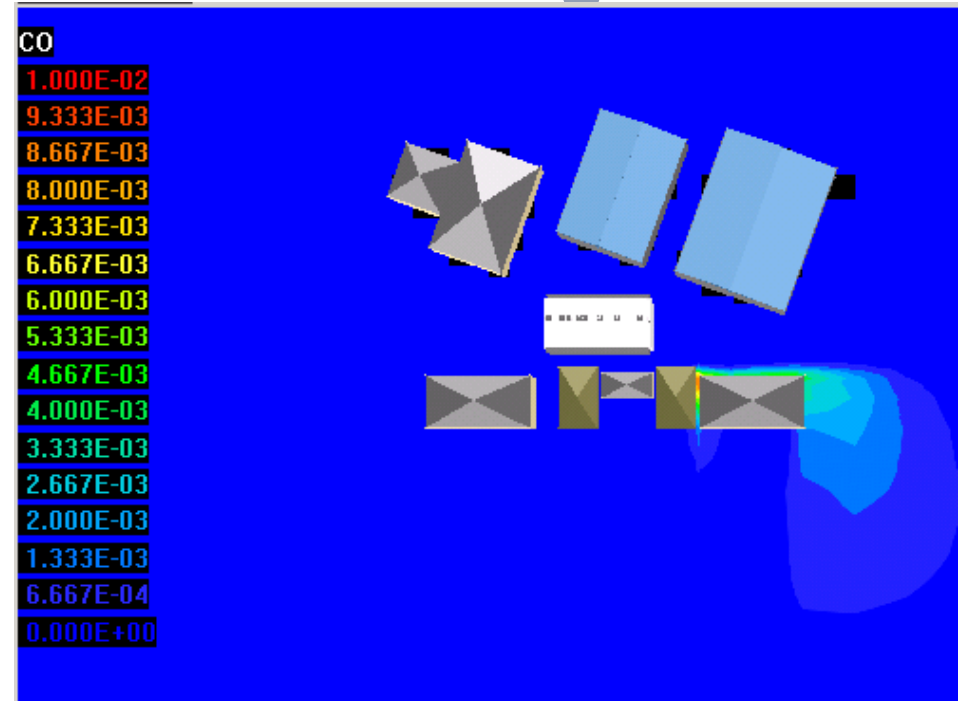
South-Westerly  
Wind Conditions

Phoenics User Conference on CFD May 2004



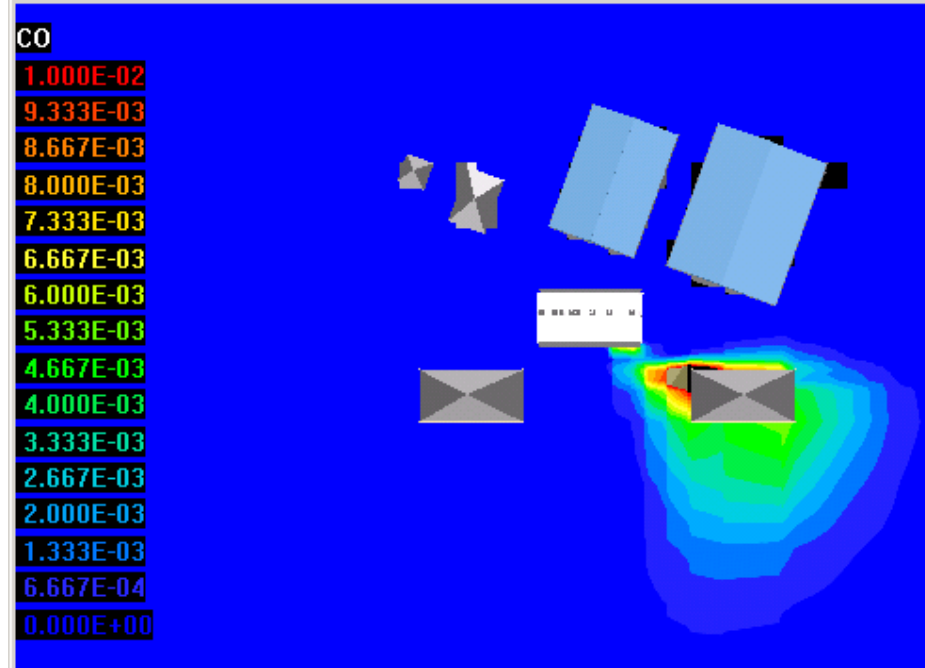
# CFD Pollutant Dispersion Results

- **concentration profile in a horizontal section at the chest level.**
- **Downwash effect**



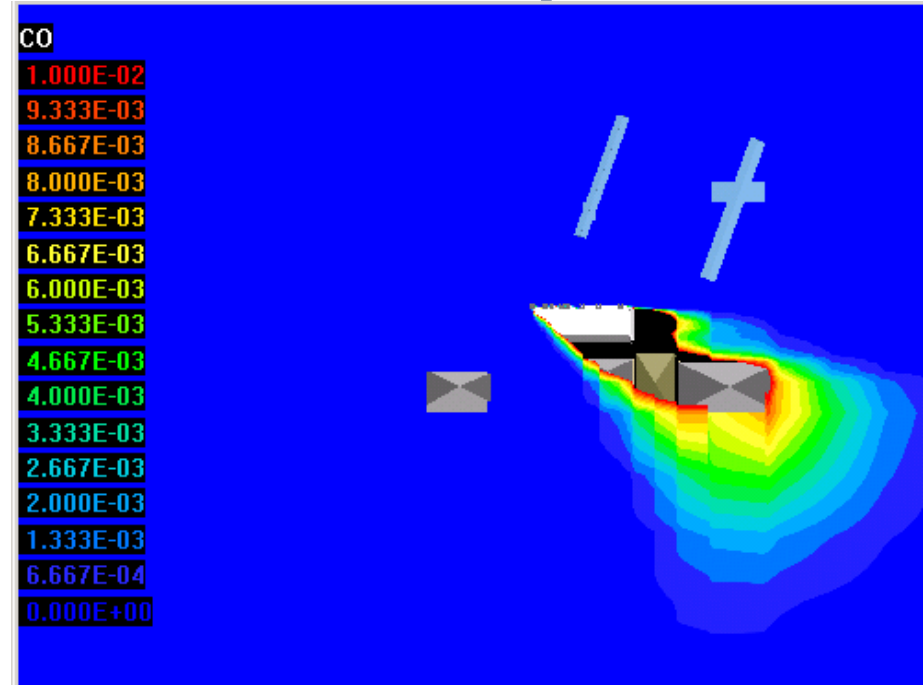
# CFD Pollutant Dispersion Results

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.



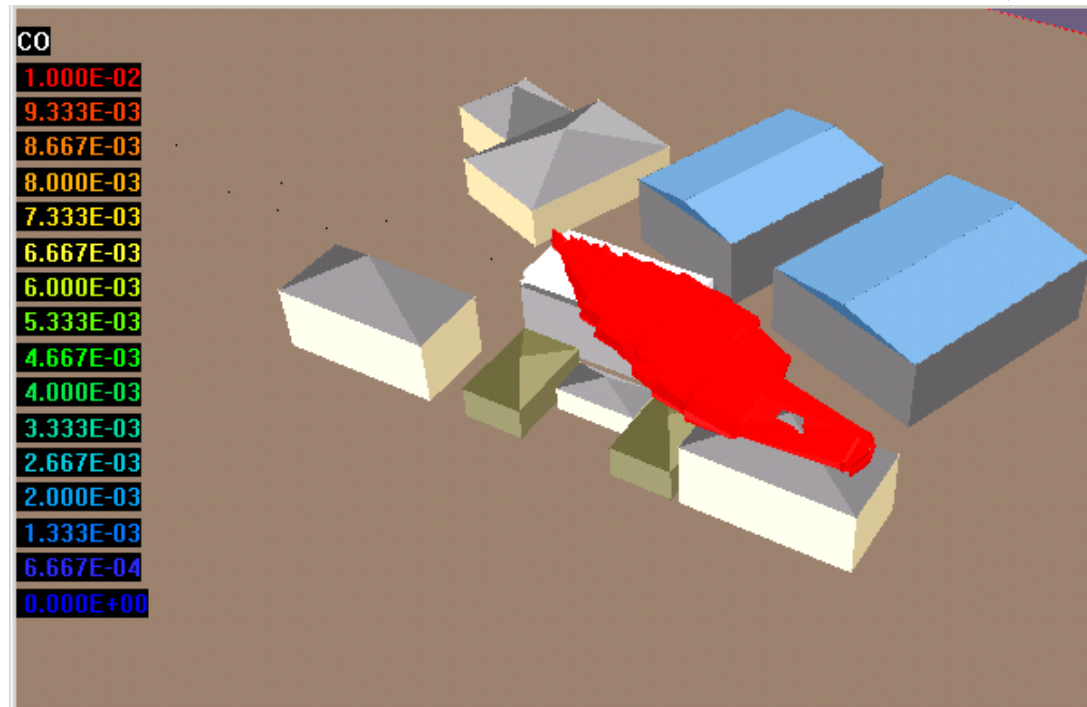
# CFD Pollutant Dispersion Results

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.



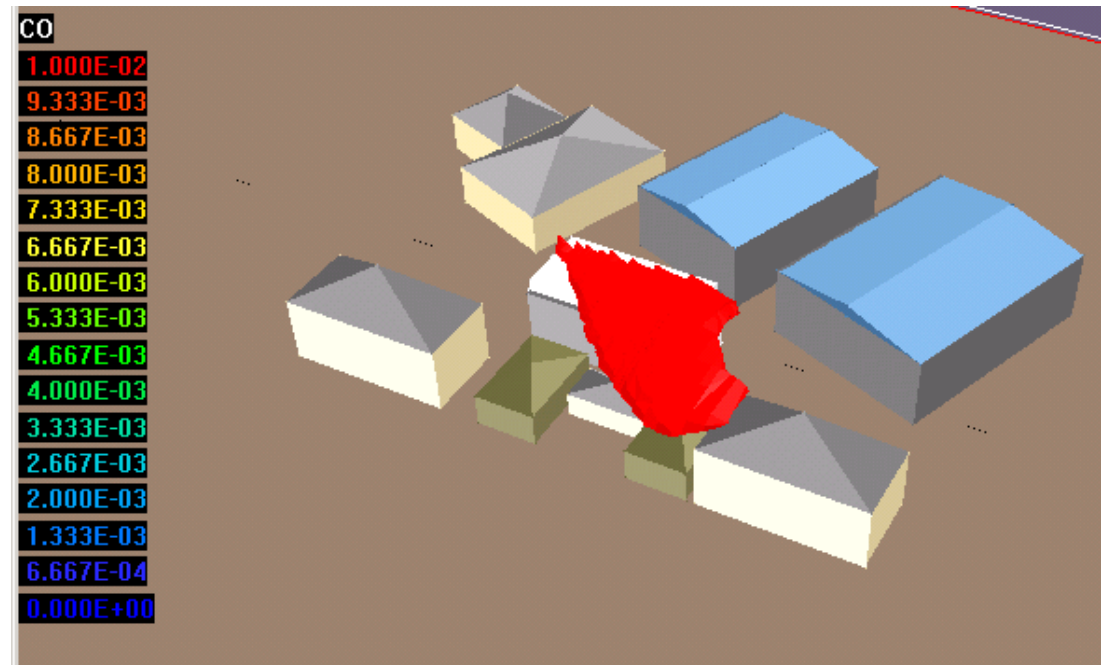


# CFD Pollutant Dispersion Results



**Concentration Profile- Iso-surface of 1%  
Source Concentration – Pollutant  
Slightly Heavier Than Air**

# CFD Pollutant Dispersion Results

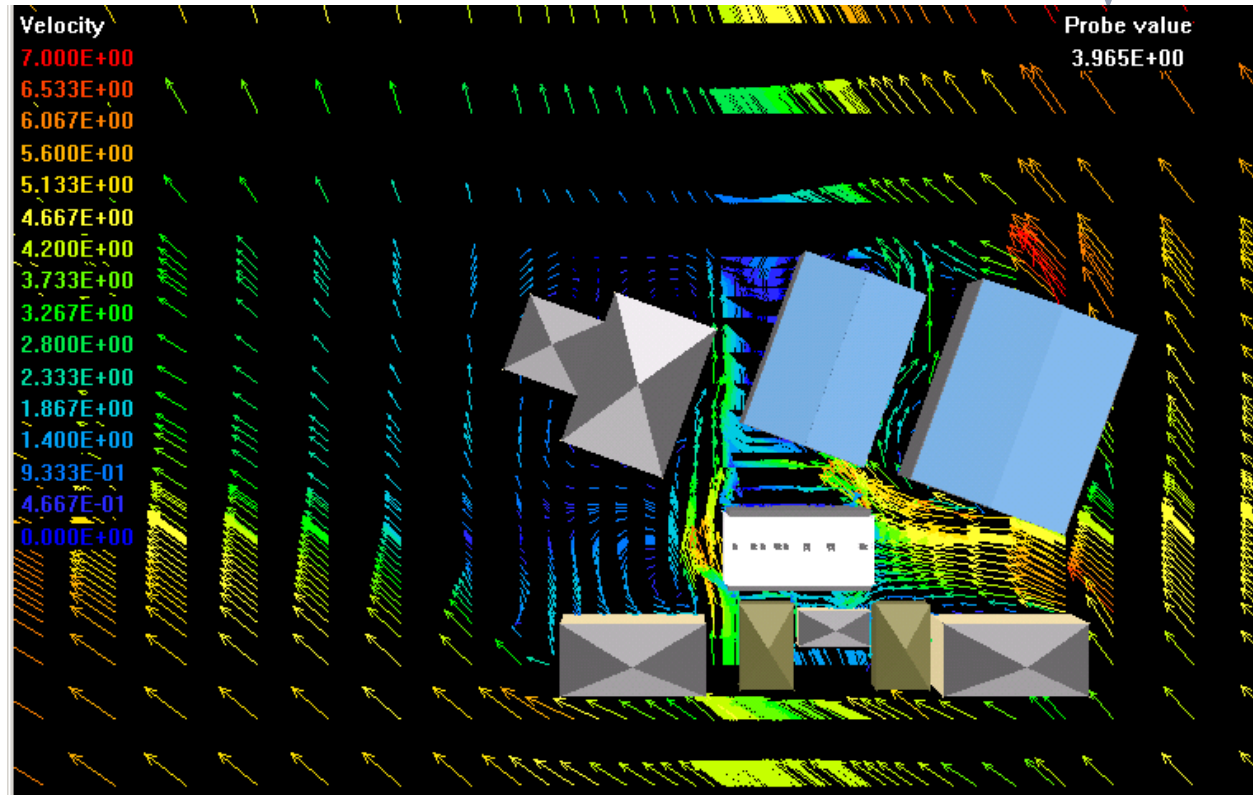


**Concentration Profile- Iso-surface of 1%  
Source Concentration – Pollutant of a  
Similar Density Than That of Air Was  
Used**

Phoenics User Conference on CFD May 2004



# CFD Flow Results

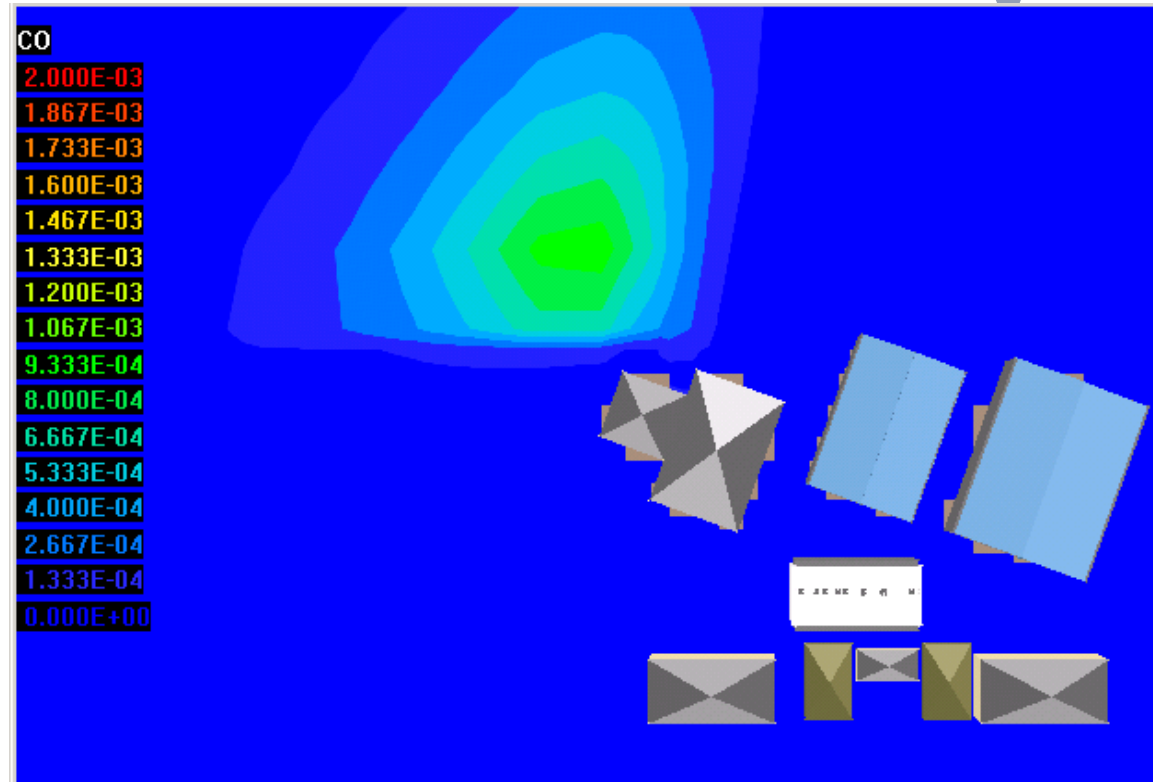


North-Easterly  
Wind Conditions

Phoenics User Conference on CFD May 2004

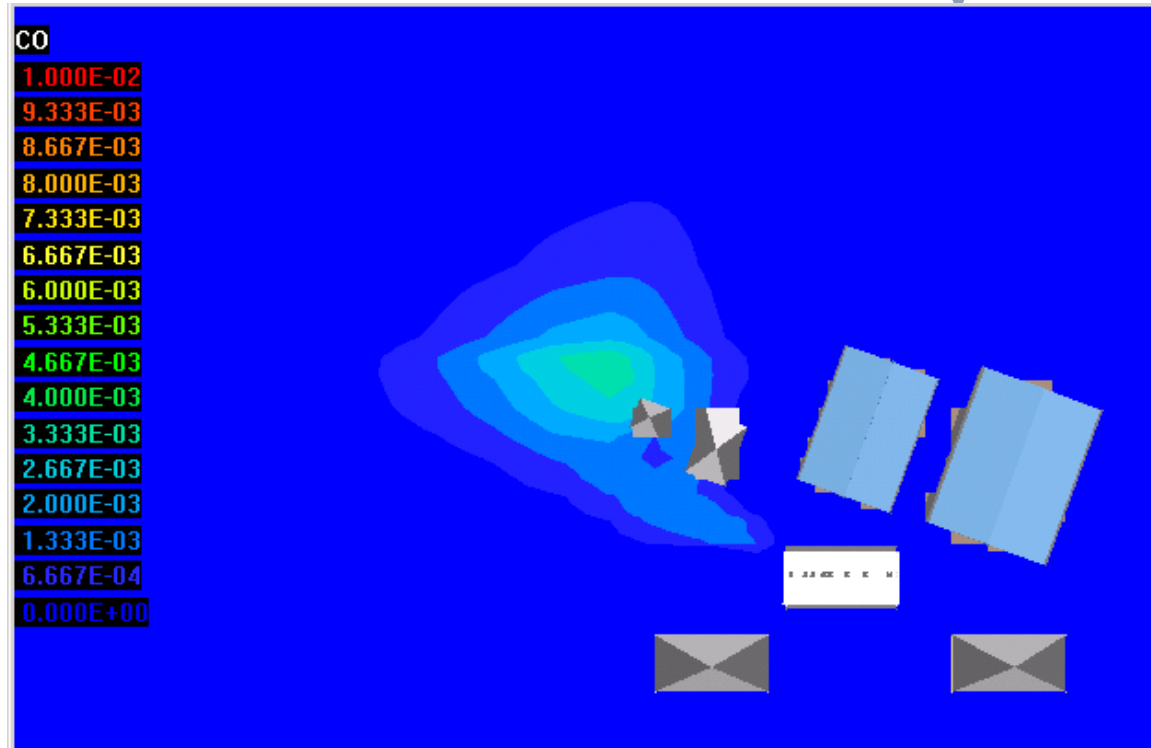


# CFD Pollutant Dispersion Results



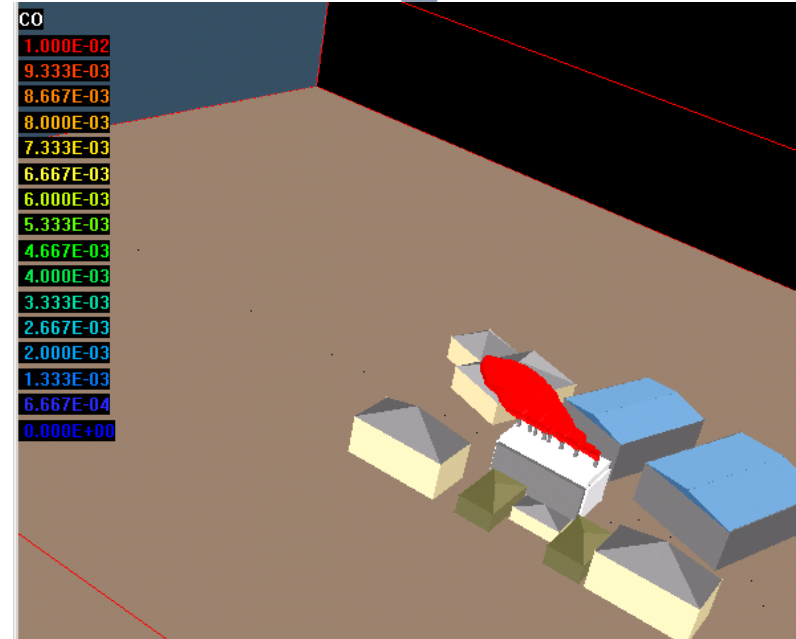
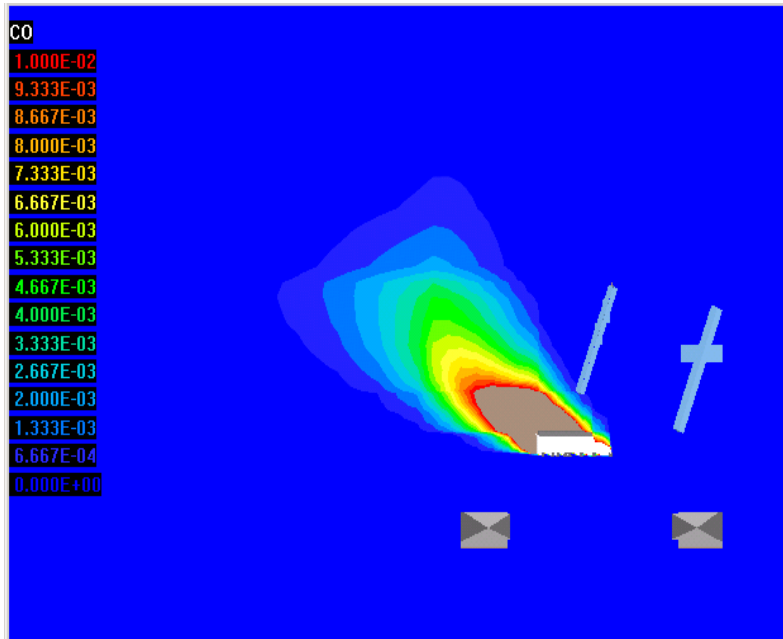
**Concentration Profile in Horizontal  
Section at 1.5 m Elevation**

# CFD Pollutant Dispersion Results



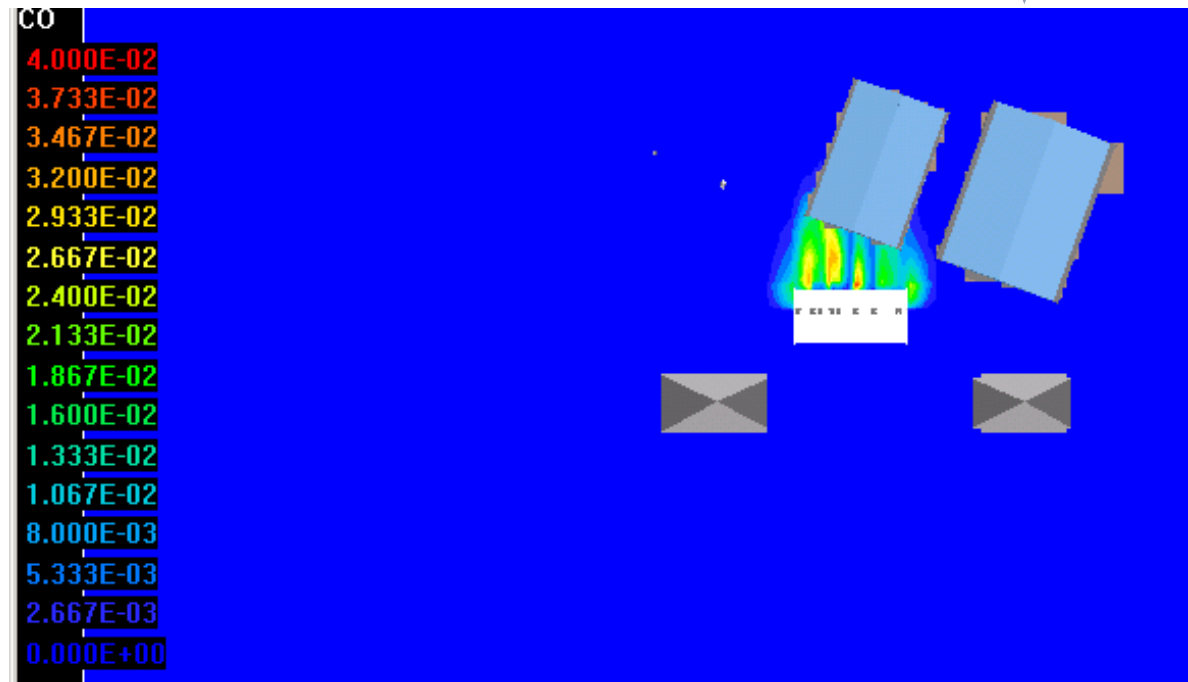
**Concentration Profile in Horizontal  
Section at 15 m Elevation**

# CFD Pollutant Dispersion Results



**North-Easterly Wind Condition**

# CFD Pollutant Dispersion Results



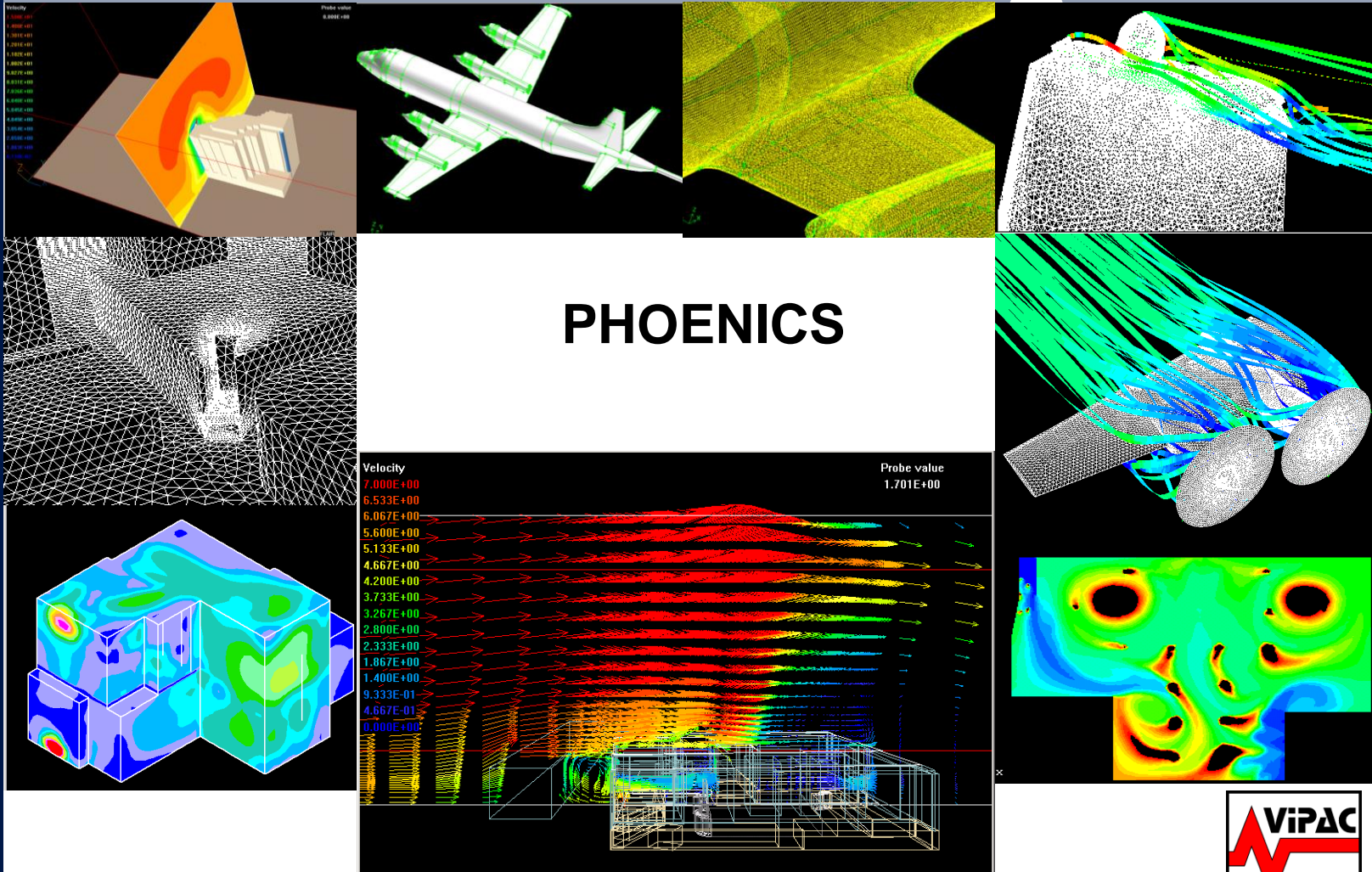
**Calm Wind Condition – near  
the roof of the building**

# 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- $\epsilon$  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 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.



# Vipac Engineers & Scientists Ltd



Phoenix User Conference on CFD May 2004

