CIBSE/ASHRAE Meeting

CFD Grows Up!

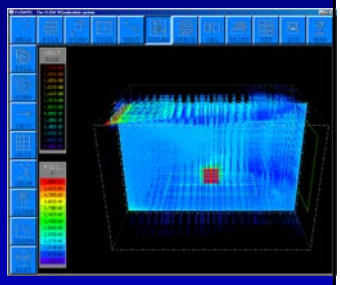
Martin W. Liddament
Ventilation, Energy and Environmental Technology
(VEETECH Ltd)

10th December 2003

What is Computational Fluid Dynamics?

CFD is a numerical means by which:

- Air Flow Distribution;
- Flow Velocity;
- Temperature Distribution;
- Contaminant Distribution;
- Fire and Smoke Movement.

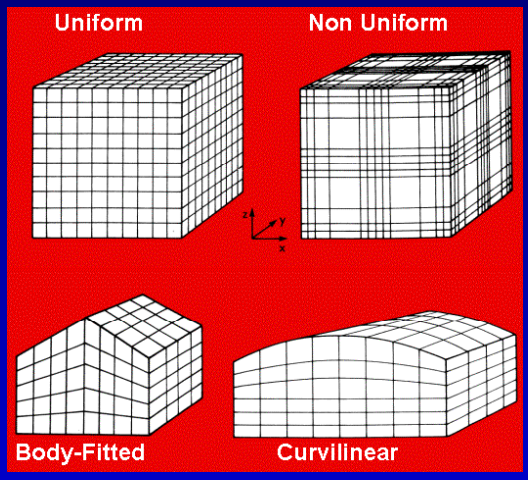


can be predicted within a space.

CFD can also be used to predict outdoor flow patterns

How does CFD Work?

A space is divided into a series of control volumes in which the equations governing air flow, thermal transport and pollutant transport are solved



Assembling the Governing Equations

Air Flow

- ***Laminar Flow*** (e.g. displacement ventilation)
  - Navier Stokes Equations ;

- ***Turbulent Flow*** **(e.g. mixing diffusers)**
  - ***k*-ε Models** (Turbulent diffusivity expressed in terms of kinetic energy of turbulence and the dissipation rate of kinetic energy of turbulence.
  - **LES Large Eddy Simulation** ***(Characteristics of large eddies incorporated into grid (has proved successful but not yet in common use. Requires significant processing time)***

Assembling the Governing Equations

Driving Forces

•*Forced Convection* (mechanical flow, wind driven, circulating fans)
- Isothermal (uniform temperature) conditions;

•*Free Convection* (buoyancy driven e.g. 'self driven' convection currents)
- non isothermal, vertical force (rising of hot air);

•*Mixed Convection* (combined forced and free convection).

Assembling the Governing Equations

Pollutants

•*Emission / Sink Rate*

•*Location*

•*Time Dependency*

Structure of Transport Equations

$$\frac{\partial}{\partial t}(\rho\varphi) + div(\rho\bar{V}\varphi - \Gamma_{\varphi} grad\,\varphi) = S\varphi$$

Unsteady Term (i.e. Time Dependent)	Convection Term	Diffusion Term	Source Term

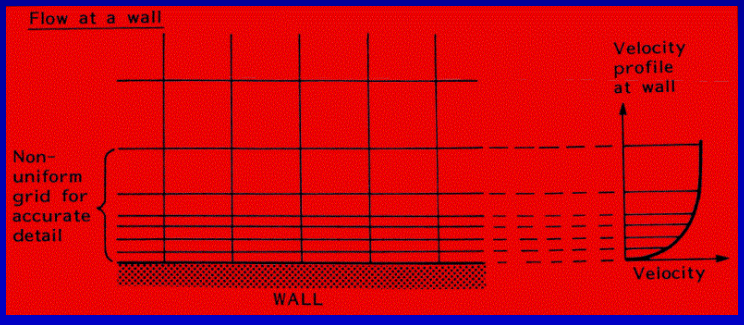
Dealing with Solid Boundaries

Flow:

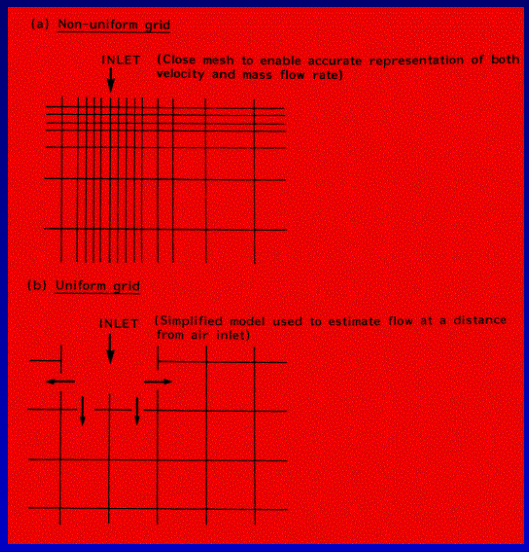
- Boundary Layer Theory;
- No Tangential Flow
- Turbulent/Laminar (Local Re No)

Thermal

- Heat Transfer Coefficient;
- Temperature;
- Grid Spacing 5mm from wall;
- Grid Expansion ratio 1.5.



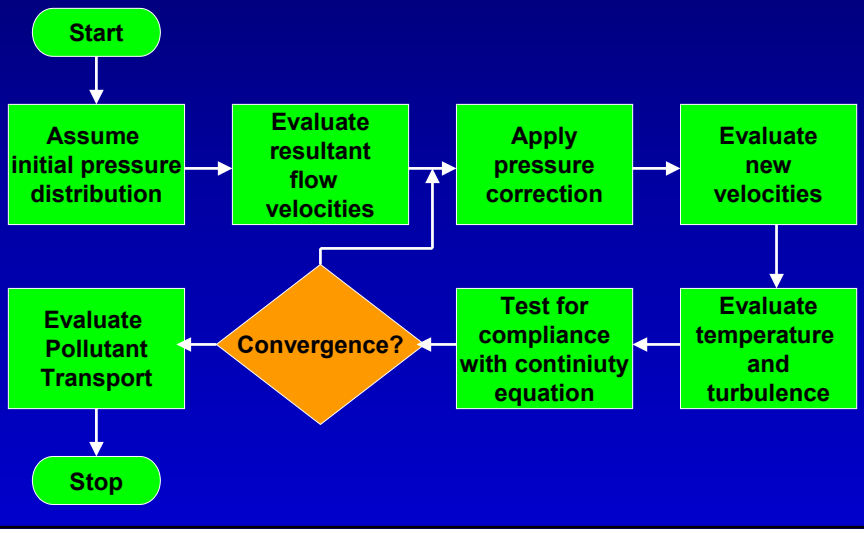
Dealing With Inlets



Must accurately reflect:
- **Air velocity;**
- **Volume flow rate;**
- **Turbulence**

Solution of Transport Equations

SIMPLE – Semi Implicit Method for Pressure Linked Equations
(Developed by Patankar and Spalding at Imperial College ~ 1970's)



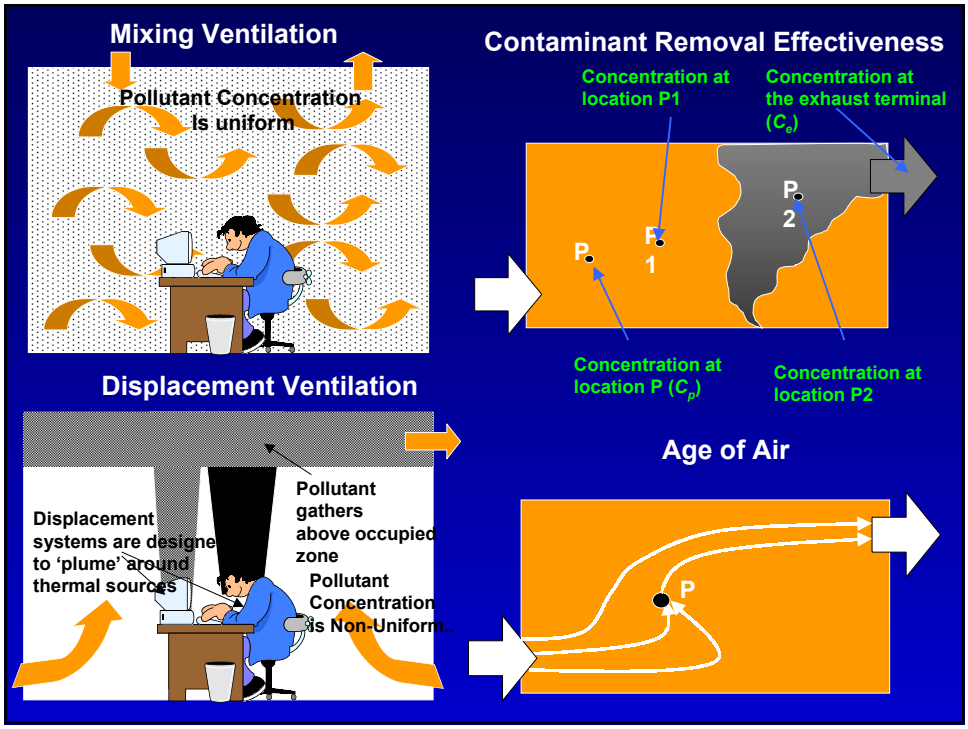
Other Issues

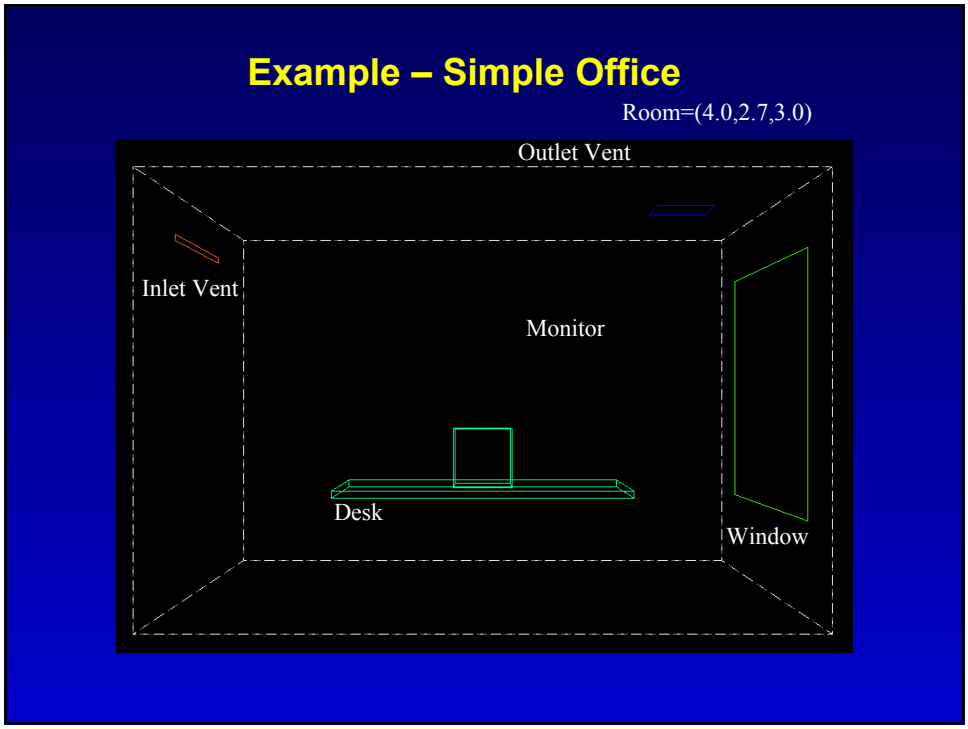
Convergence Criteria – Mass balance error e.g. < 1%

Solution Time – Can be hours or even days

Grid Dependency – Solve for more than one grid density

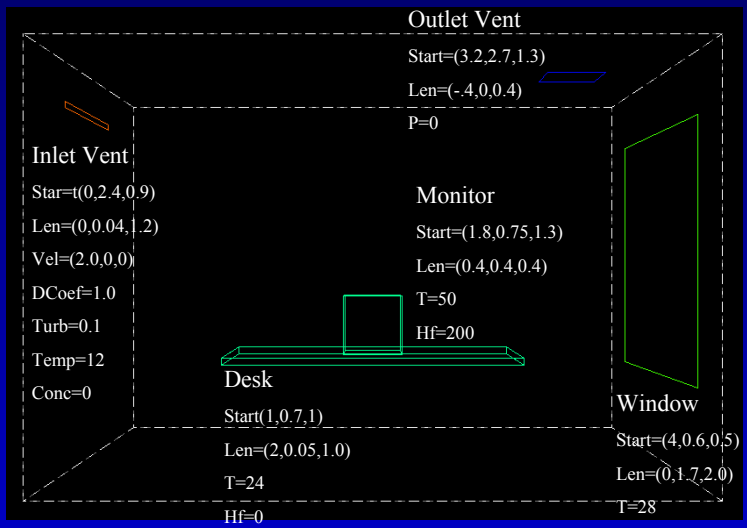
Some Examples


Mixing Ventilation
Pollutant Concentration Is uniform
Contaminant Removal Effectiveness
Concentration at location P1
Concentration at the exhaust terminal (C_e)
P 2
P 1
P
Concentration at location P (C_p)
Concentration at location P2
Displacement Ventilation
Pollutant gathers above occupied zone
Displacement systems are designe to 'plume' around thermal sources
Pollutant Concentration is Non-Uniform..
Age of Air
P

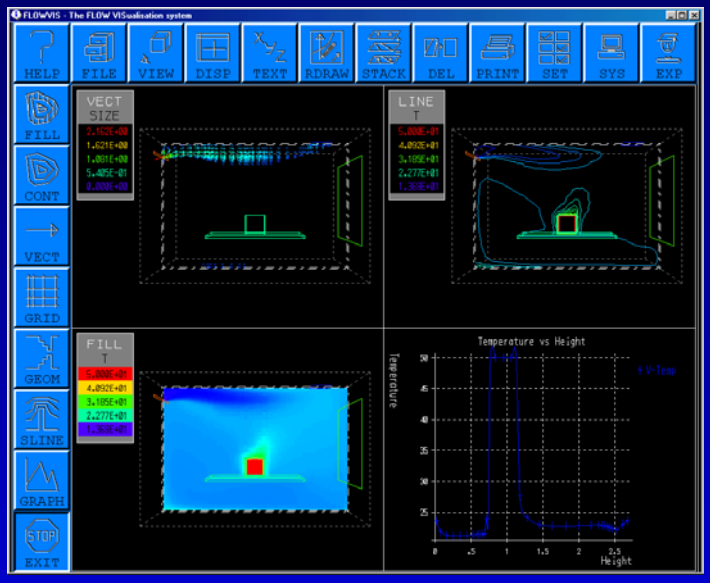

Example – Simple Office
Room=(4.0,2.7,3.0)
Outlet Vent
Inlet Vent
Monitor
Desk
Window

Example – Building Up a Network

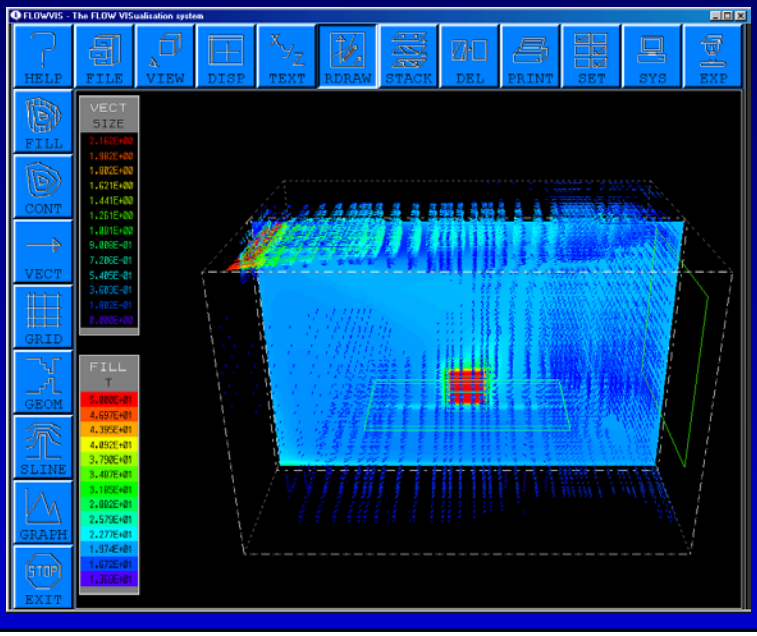
Room=(4.0,2.7,3.0)



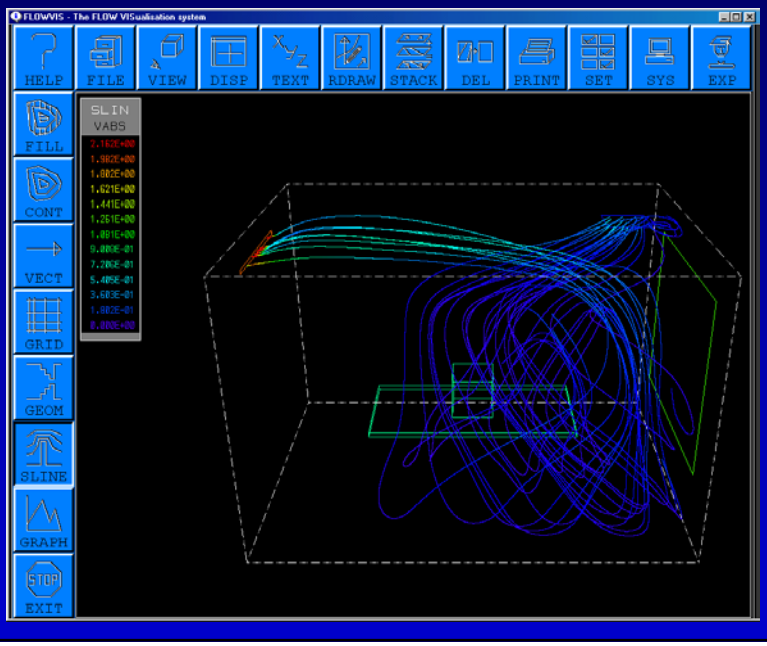
Example – Typical Results



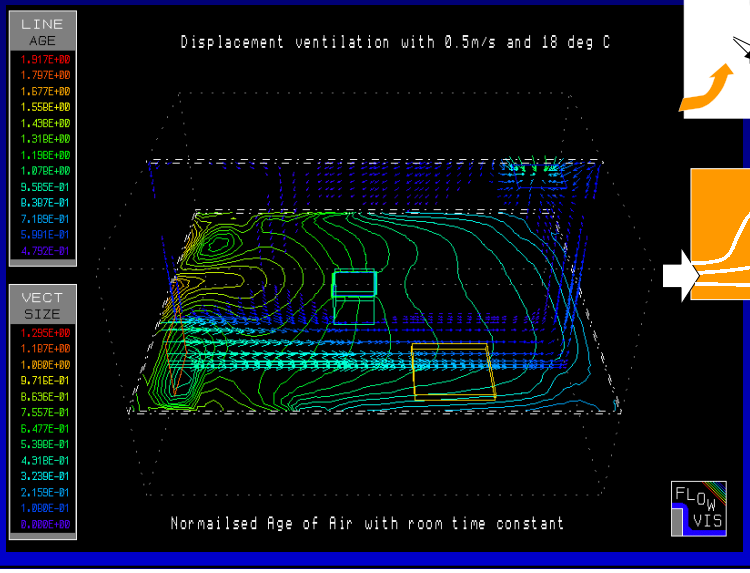
Results - 3D Visualisation



Results Streamlines

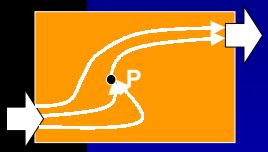


Example Displacement Ventilation

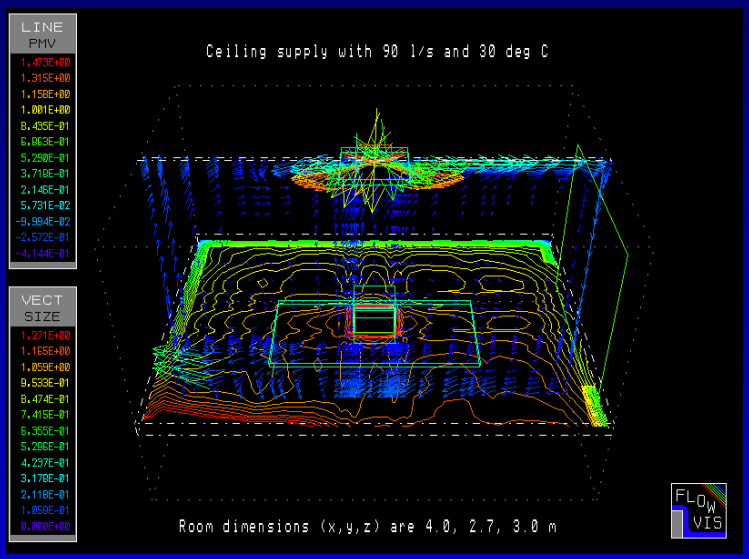


Displacement Ventilation

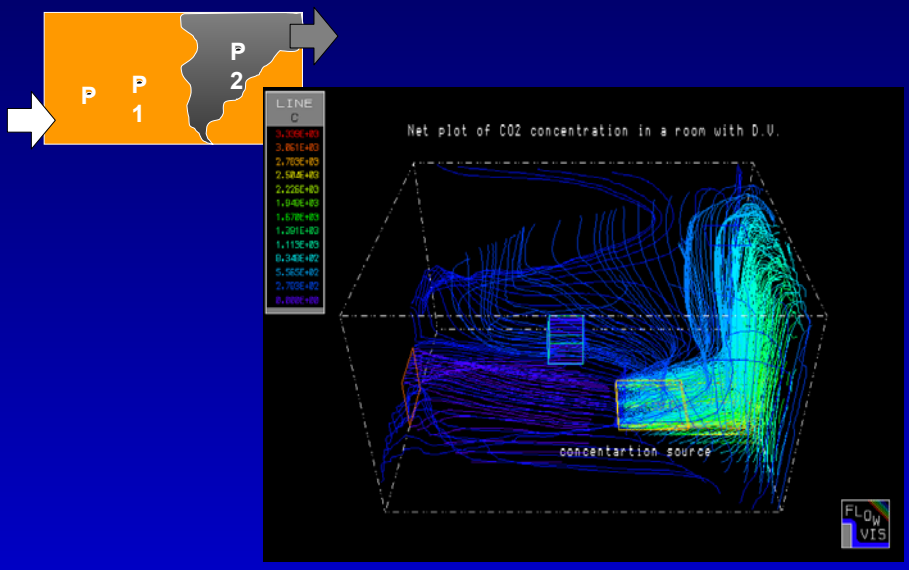
Age of Air



Thermal Comfort Prediction

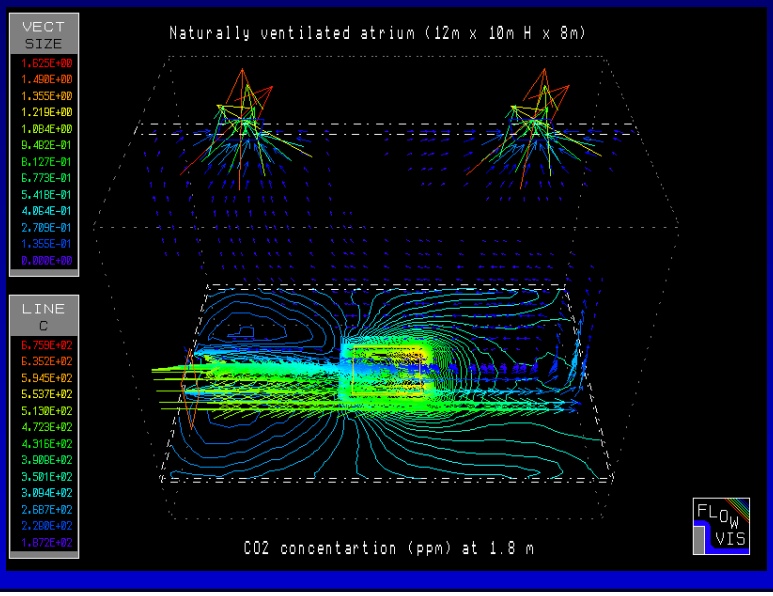


Indoor Air Quality Prediction



Natural ventilation

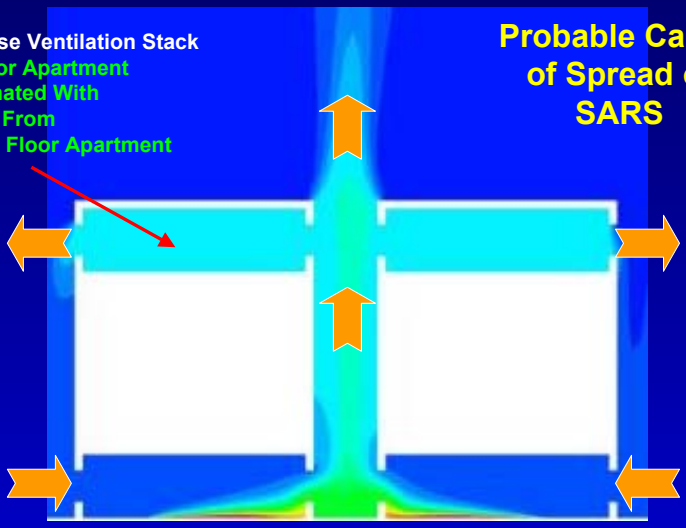
Buoyancy-driven flow in an atrium



Cross Contamination Between Apartments

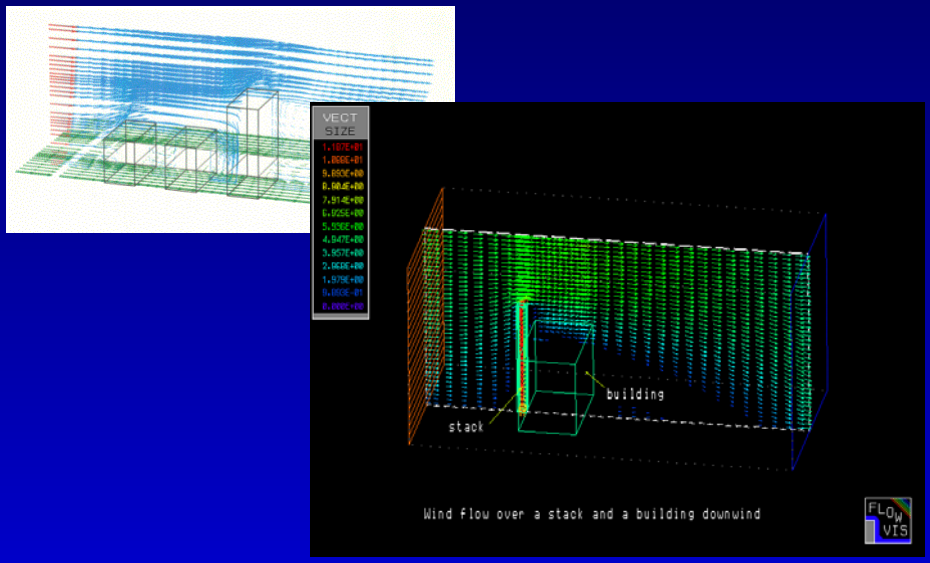
High – Rise Ventilation Stack
Top - Floor Apartment
Contaminated With
Pollutant From
Ground – Floor Apartment

Probable Cause of Spread of SARS

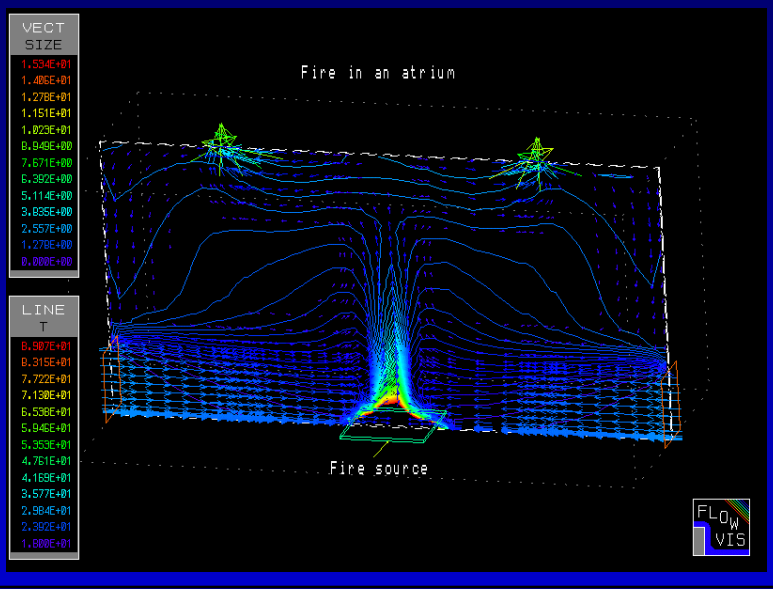


Lui
&
Moser
IJV
Feb
2003

Wind Flow over Buildings



Fire Spread



Summary

CFD is an effective tool for assessing and visualising flow related behaviour.

It can be used to analyse a wide variety of prblems including air flow, temperature and pollutant distribution.

CAUTION!

CFD is VERY Seductive.

Work must be supported by conventional analysis

Where is it Possible to get CFD Code?

Commercial Code

- FLOWVENT (Flowmerics);
- CFX (Harwell);
- PHOENIX (Cham);
- STAR-CD
- VORTEX (Reading/Greenwich Universities)*.

Public Domain

- TEACH (forerunner of PHOENIX);
- EXACT3 (from NIST in US)*.

* On CD-ROM

CD to Take Away

Full Version of VORTEX CFD *(Valid to 15th Jan 2004)*

FORTRAN Source Code: EXACT3 *(3-D CFD Simulation Code)*

Some Copies of:

The International Journal of Ventilation

for light Christmas Reading

THE END

For more information contact:
Mliddament@veetech.co.uk