# Analysis of Building Performance using Computational Fluid Dynamics (CFD)

Richard Chitty

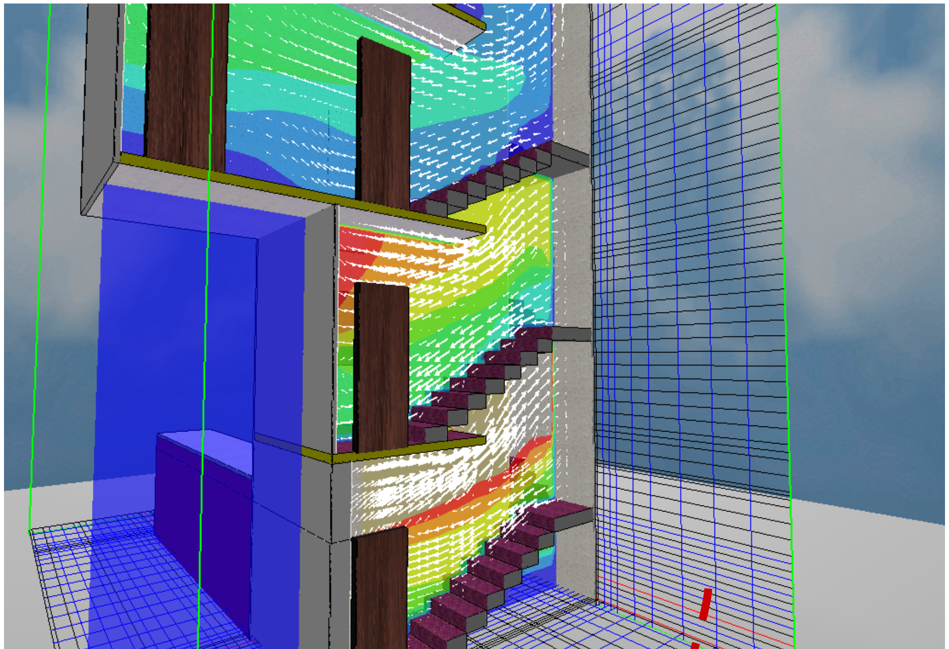
# Content

- Computational Fluid Dynamics (CFD)

- Natural Ventilation System
- Wind Application
- Fire Safety Application

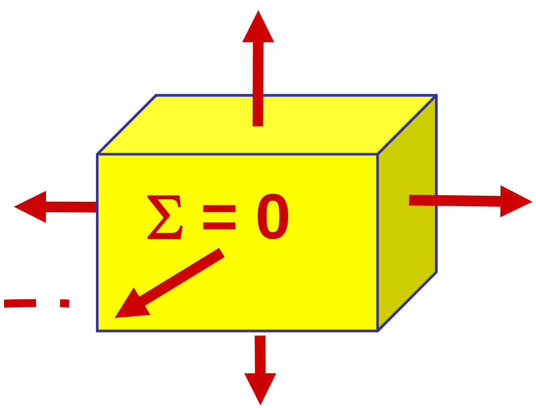
- Conclusion

# CFD modelling - in one slide



- Discretise geometry
- Select physical sub-models
- Apply boundary conditions
- Solve coupled equations
- Process solution data

conservation equations imposed at each mesh element



# CFD study of a Novel Naturally Ventilated Building

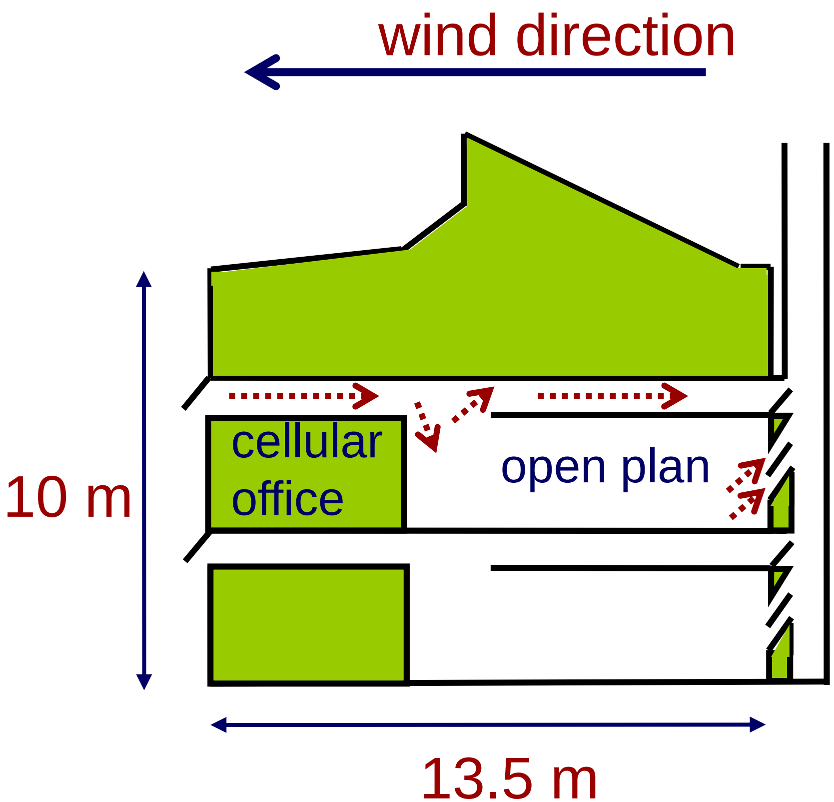
BRE Environmental Office
Opened 1997

# Ventilation system

- Cooling is achieved by:
  - Cross ventilation
  - Groundwater cooling

- Exposed ceiling slab with air channels
- 5 external ventilation shafts

- CFD simulations were undertaken to study the environmental conditions in the first-floor open plan office during a warm summer day with a light southerly breeze




- ambient air = 24°C
- wind speeds of 0.5 m/s & 1.5 m/s (at 10 m height)
- ceiling channels open to the outside
- external shaft windows open
- cooling by the ceiling slab
- 20 $W/m^2$ thermal load

# Ventilation shafts

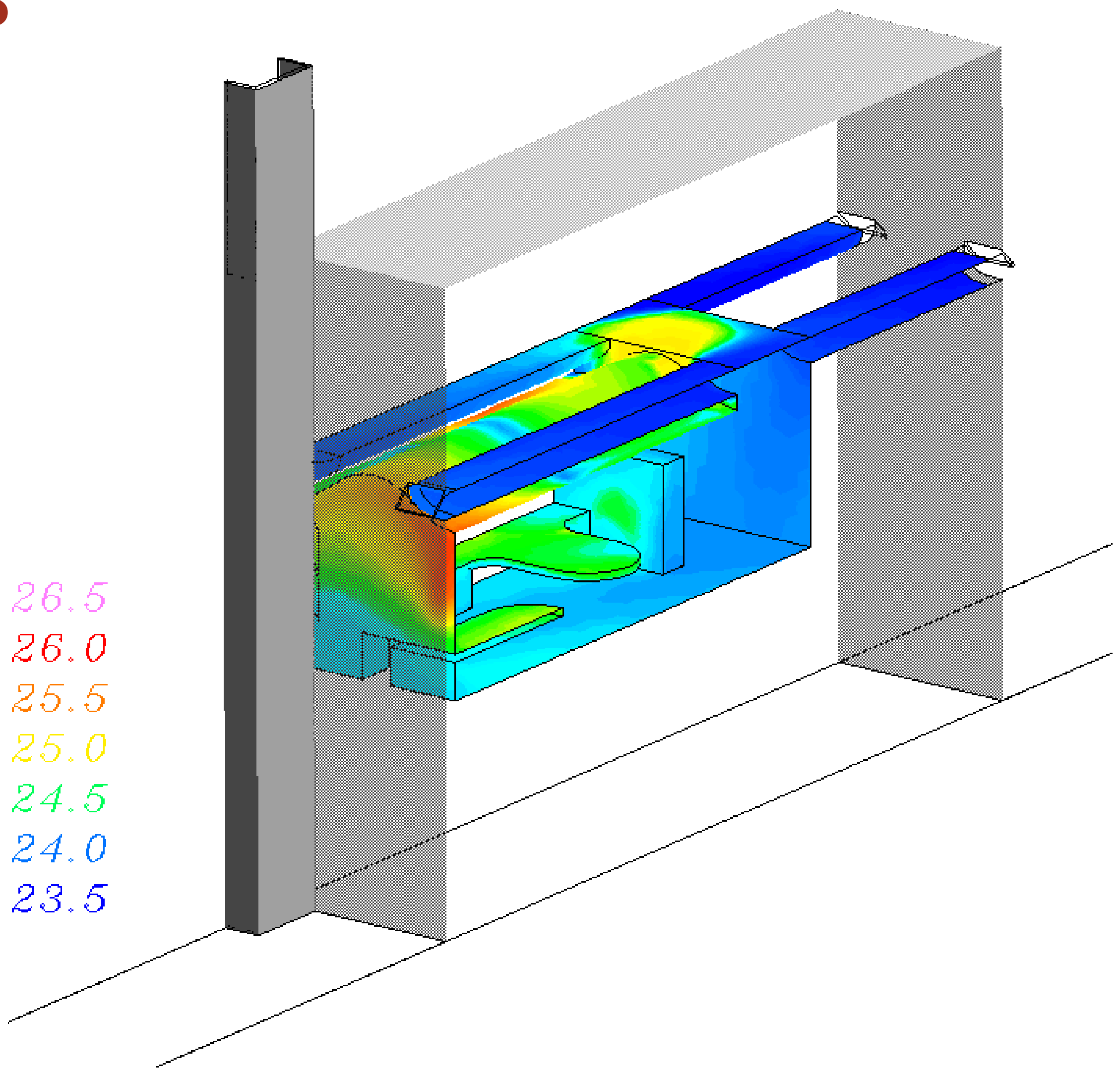
Air is drawn from the office space and upwards through the shaft with the assistance of:

- wind flow across the top
- solar warming of the glass panels
- a low power fan inside the shaft

Little understanding of their actual effect when the building was constructed

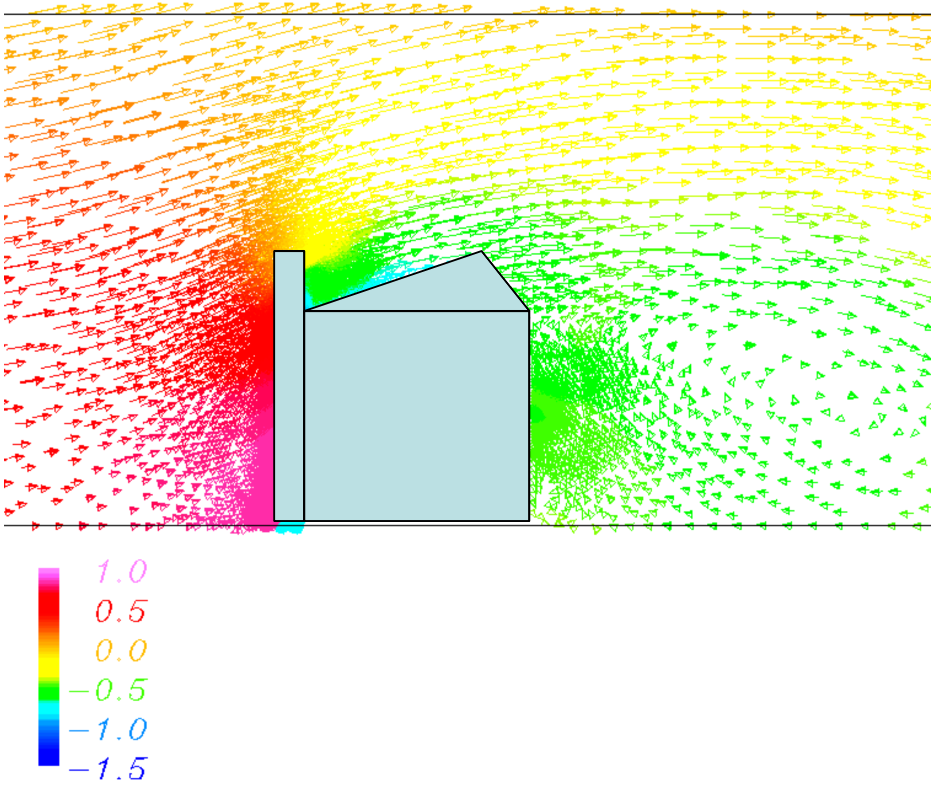
# CFD simulations

- A ‘slice’ of the building and external atmosphere has been simulated, allowing
- fine numerical resolution
- interaction of the breeze with the building to be modelled, removing the need for assumed pressure coefficients.



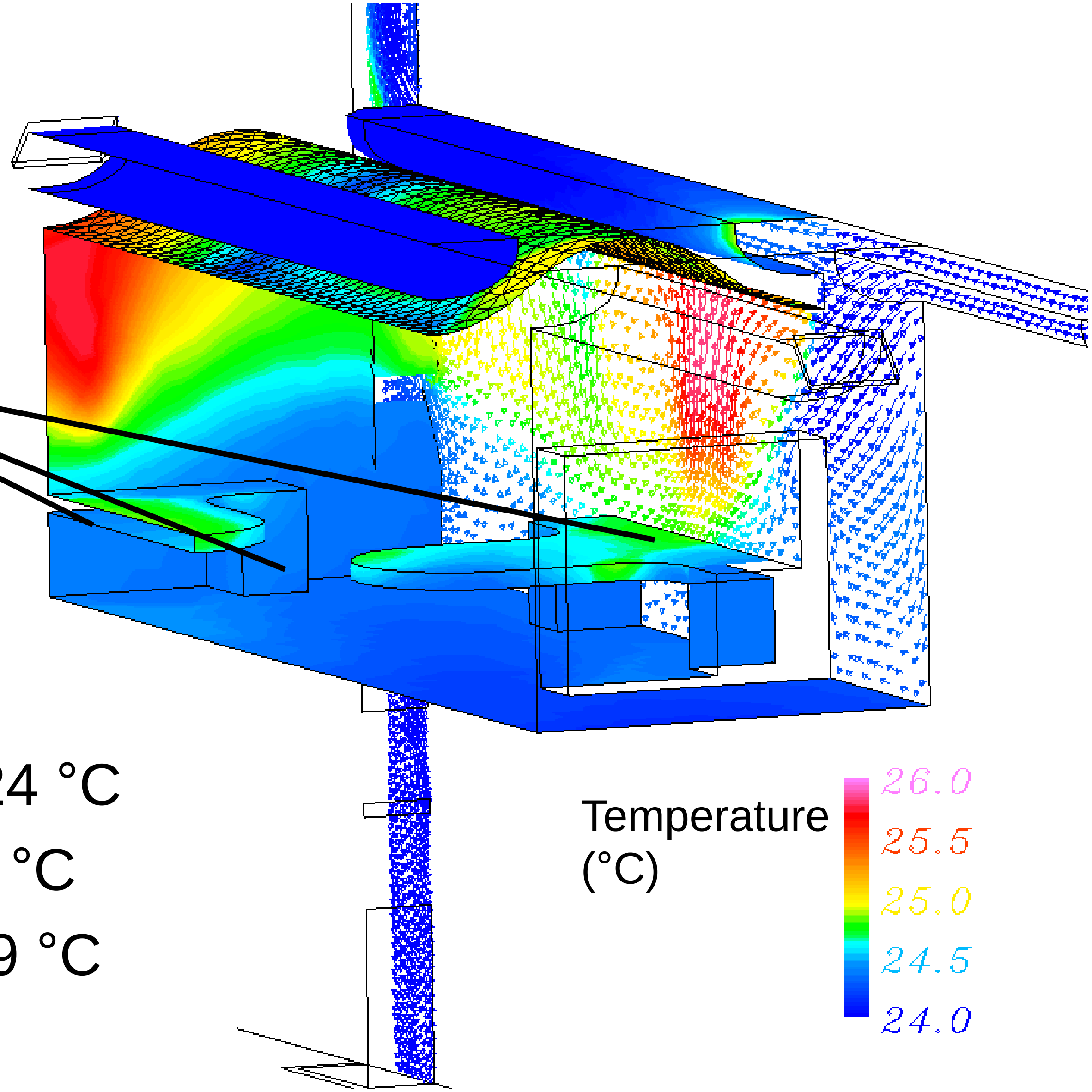
# External breeze

An inlet boundary condition placed upstream to provide a logarithmic wind profile



# Unstructured mesh of 470000 elements

volume heat sources



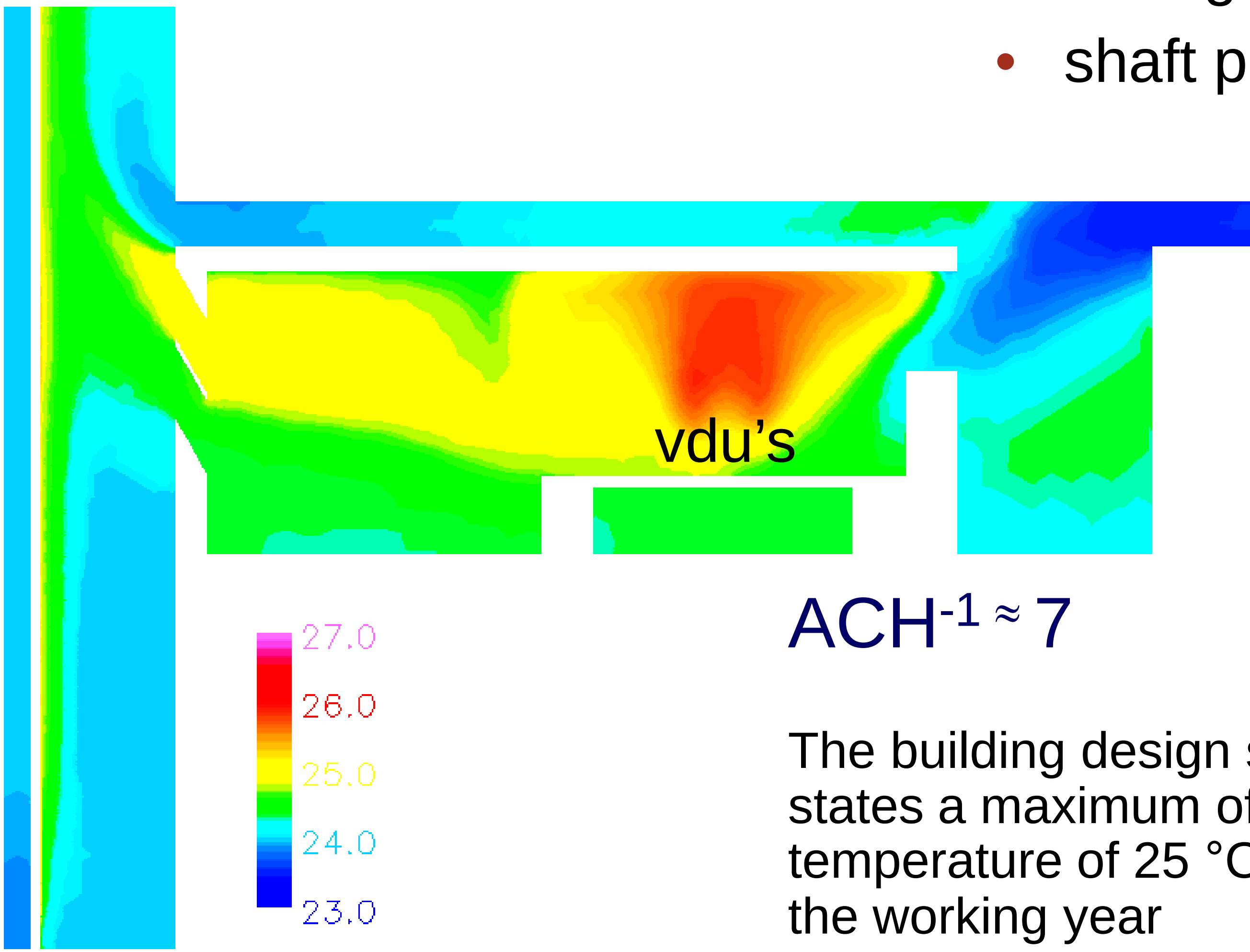

- 1.5 m/s breeze
- outside temp at 24 °C
- ceiling slab at 21 °C
- shaft panels at 29 °C

# CFD Simulations using CFX-5

- Fully coupled momentum-pressure solver, eliminating need for pressure-correction
- Unstructured numerical mesh of triangular surface elements, boundary layer prisms and tetrahedral volume elements
- Symmetry boundary conditions either side of the modelled ‘slice’
- ‘Standard’ k-$\varepsilon$ turbulence model
- Volume heat sources to represent vdu’s and people
- Boussinesq buoyancy approximation

# CFD Simulations

- 1.5 m/s breeze
- outside temp at 24 °C
- ceiling slab at 21 °C
- shaft panels at 29 °C



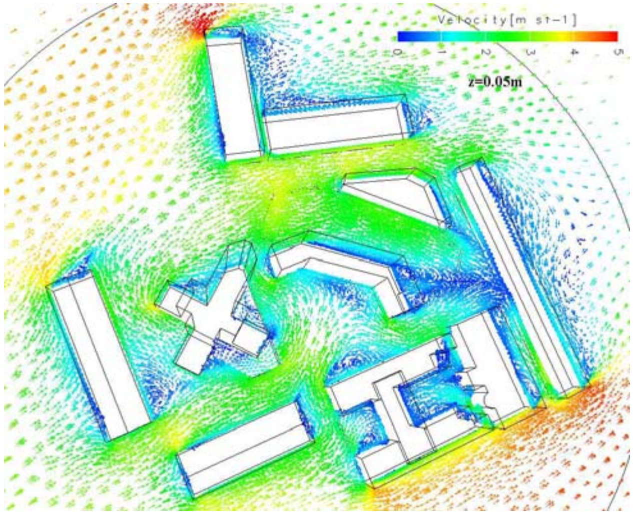

$ACH^{-1} \approx 7$

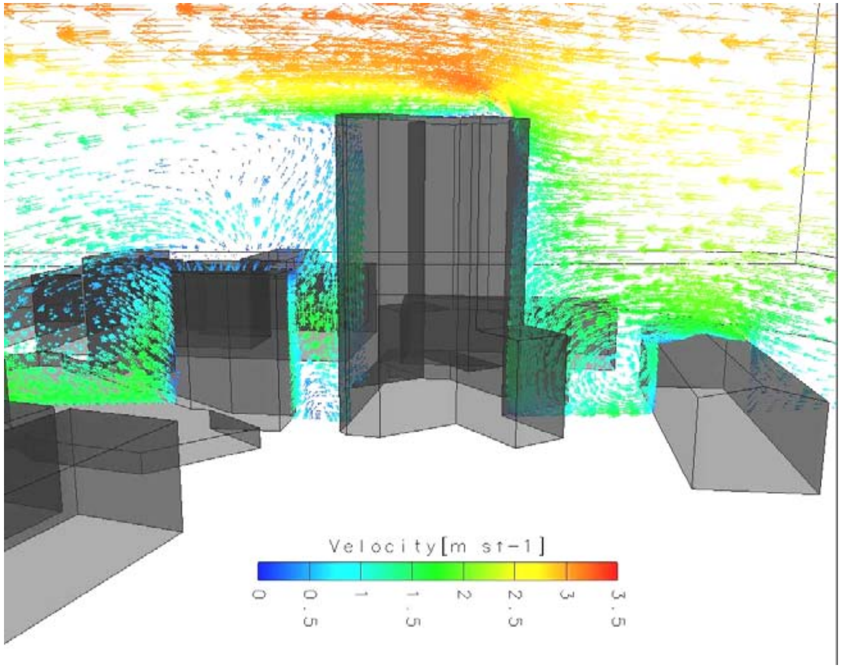
The building design specification states a maximum office temperature of 25 °C for 95 % of the working year

# Summary of conclusions

- A parametric CFD analysis has provided additional insight into the summer time operation of the BRE's Environmental Building
- With a light warm breeze and a combination of cross- and external shaft ventilation, conditions inside the first floor office shown to be acceptable
- Air change rates quite high in the presence of a 1.5 m/s breeze
- Solar heating of the external shafts not critical, as assumed in the building design
- 'Hot' daytime operation would likely require trickle ventilation and, in some instances, groundwater cooling (as happens in practice)

# Wind Engineering



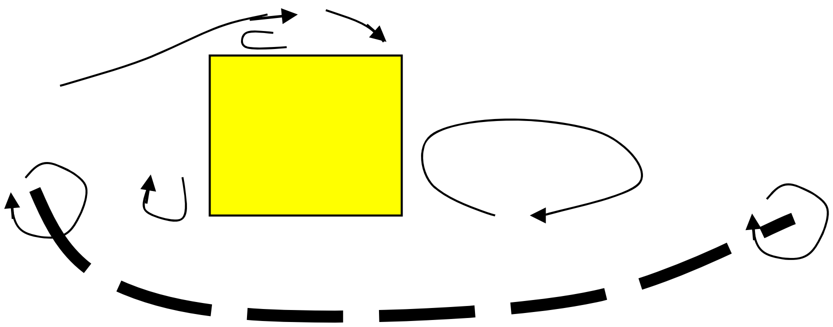



# Background

- Wind tunnel technology well established
- CFD now provides an alternative/complimentary tool
- BUT, wind engineering community has reservations

# Bluff Body Aerodynamics

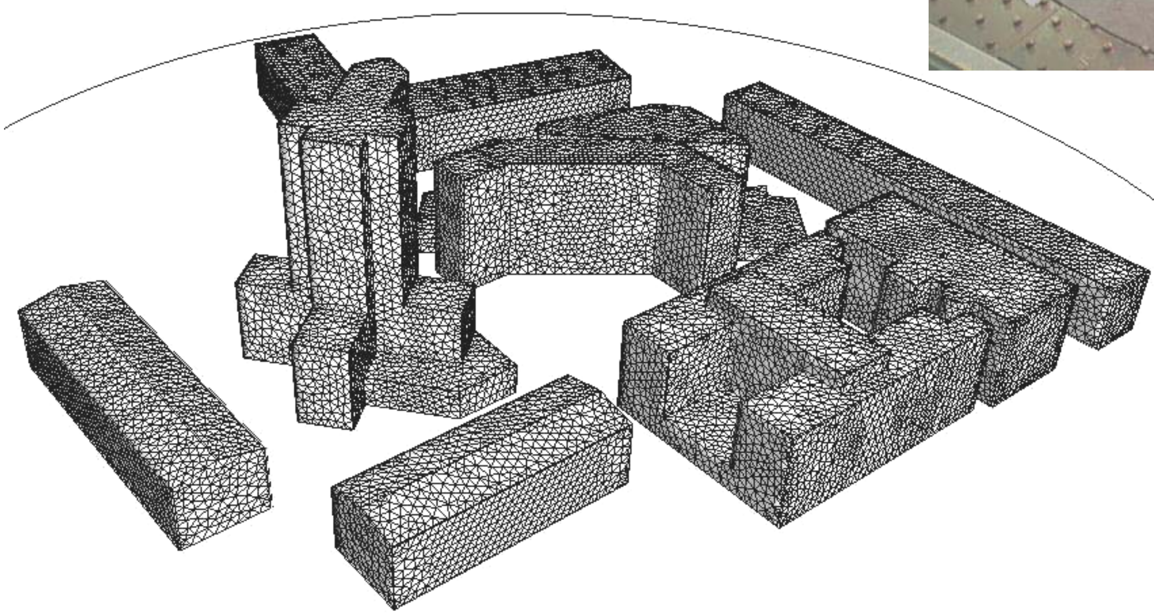
- Buildings are bluff bodies within the surface boundary layer, generating:
  - stagnation
  - separation
  - reattachment
  - vortex generation



- Flow field is inherently unsteady
  - time-averaged flow field may be quite different to the instantaneous one

# Gloucester Road Development

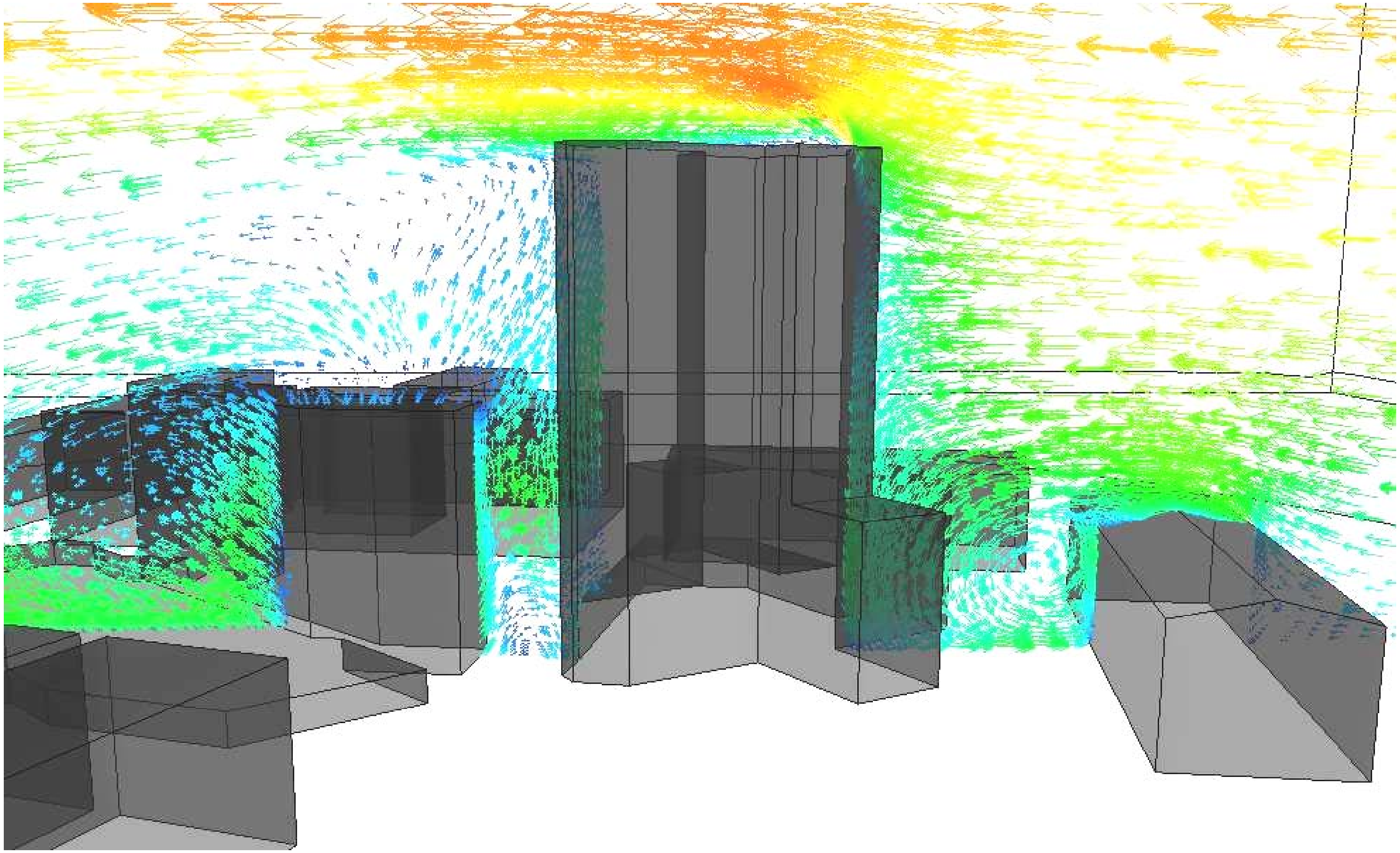
1:200 scale wind tunnel model of city centre development



1.2 million element CFX-5 model

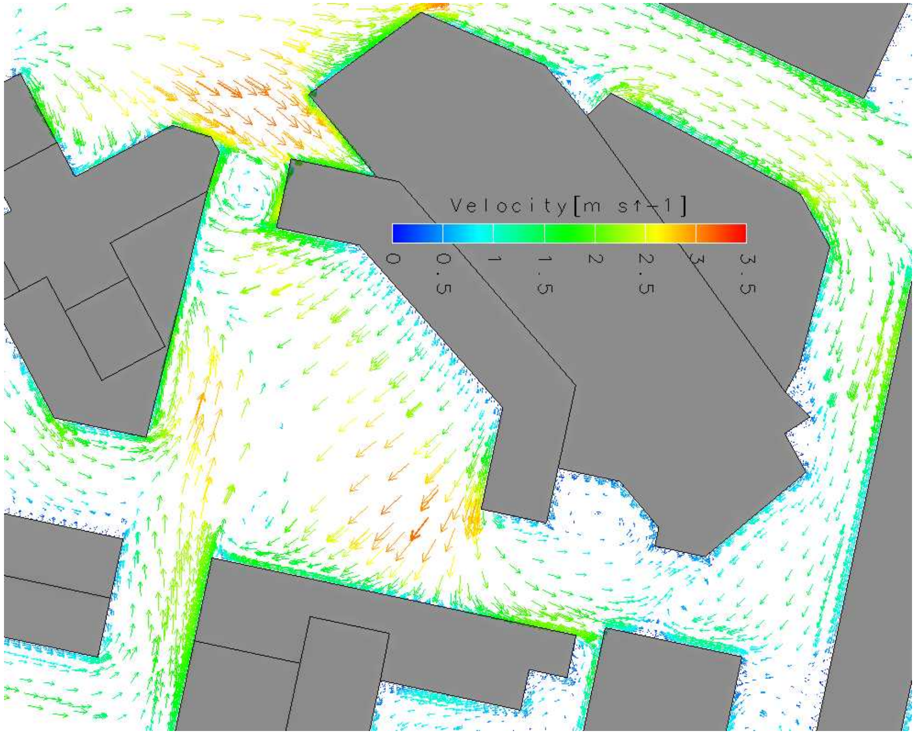
SST turbulence model

# Gloucester Road Selected Wind Direction



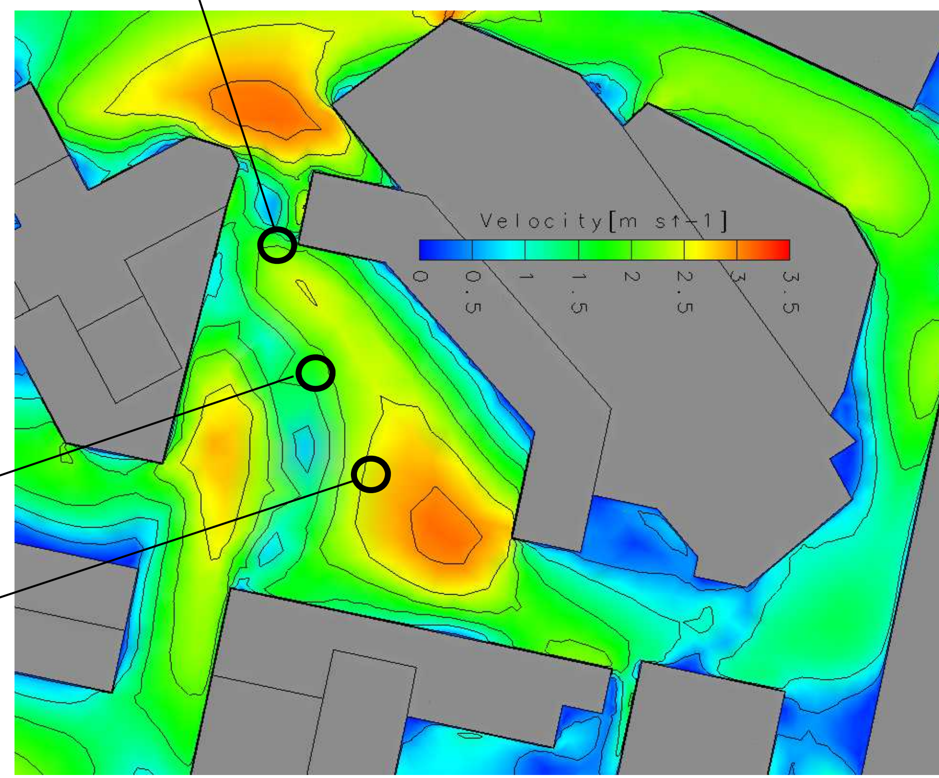
Impingement, separation, re-circulation, stagnation etc all present

# Gloucester Road Pedestrian Velocities (1.2m)




Measured v ~ 1.5 m/s

CFD v~2.75m/s




Measured v ~ 1.9 m/s

CFD v~3.15 m/s

Measured v ~ 2.1 m/s

CFD ~2.5m/s

# Comments

- Wind tunnels still have an important role
  - unsteady phenomena
  - boundary layer generation
- COST best practice guide

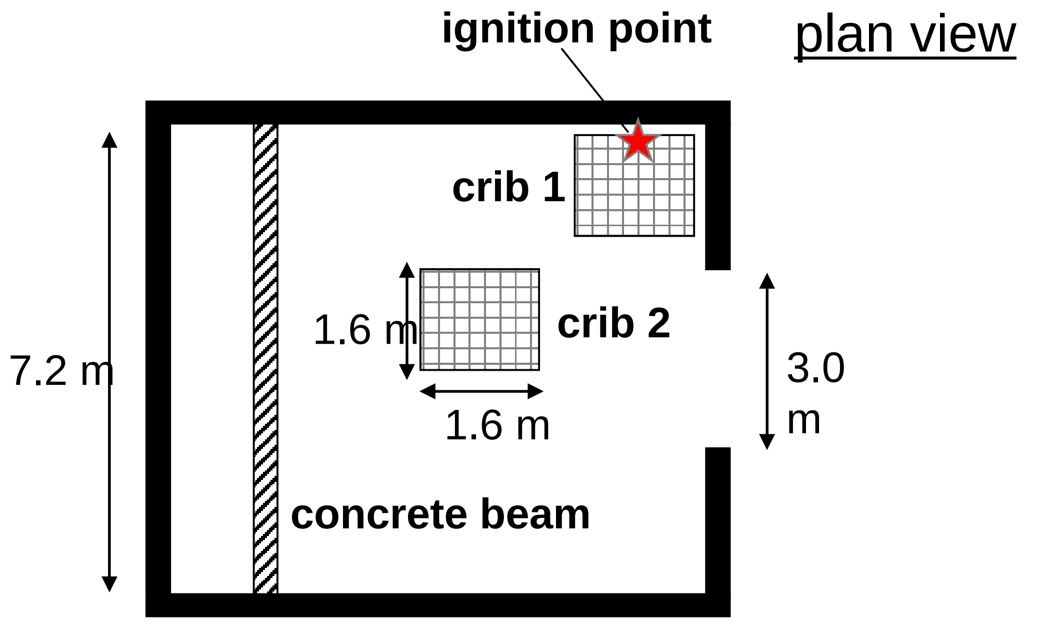
# “Blind” CFD simulation of a fire experiment

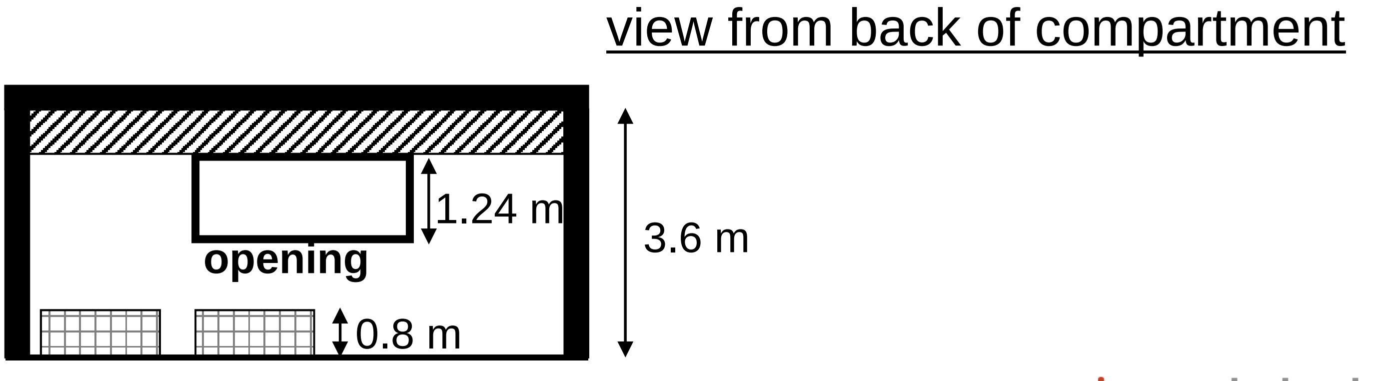
- Part of CIB W14 programme
  - Design specification issued
    - *experiment details*
    - *scenario to be modelled*
  - **‘Blind’** predictions made and submitted
  - Experimental measurements released
    - *comparison with predictions made*
    - *new ‘open’ predictions allowed*

# Fire experiment

- Conducted at VTT in 1980s
  - compartment with single opening
  - concrete block construction
  - wood crib fire sources
  - measurements
    - *temperature*
    - *gas species*
    - *wall fluxes*

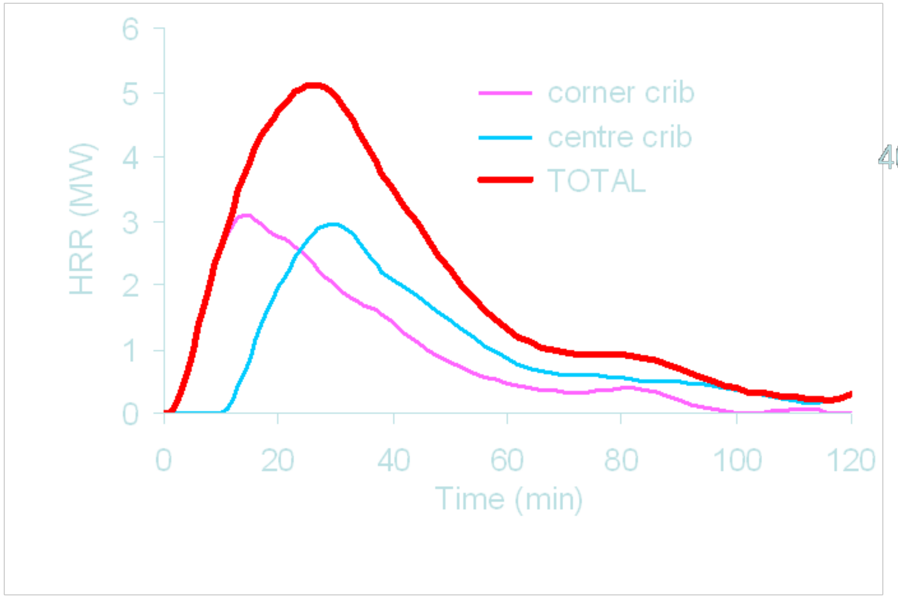
# Fire experiment

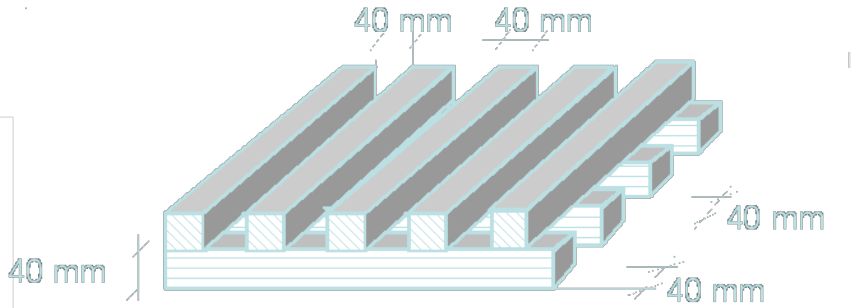


# Fire Experiment

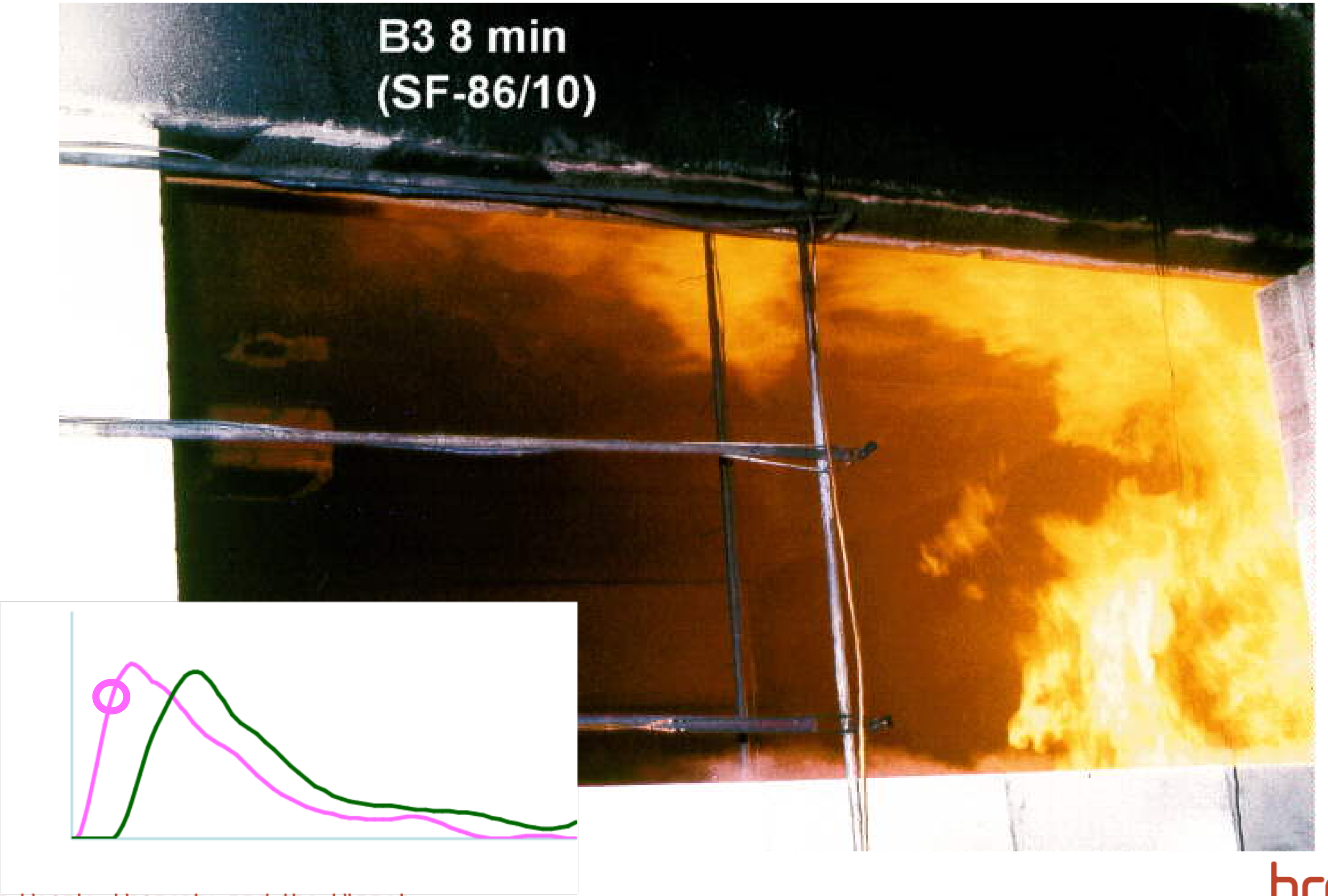
- Two softwood cribs
  - Fire peaks at 5 MW after 25 minutes



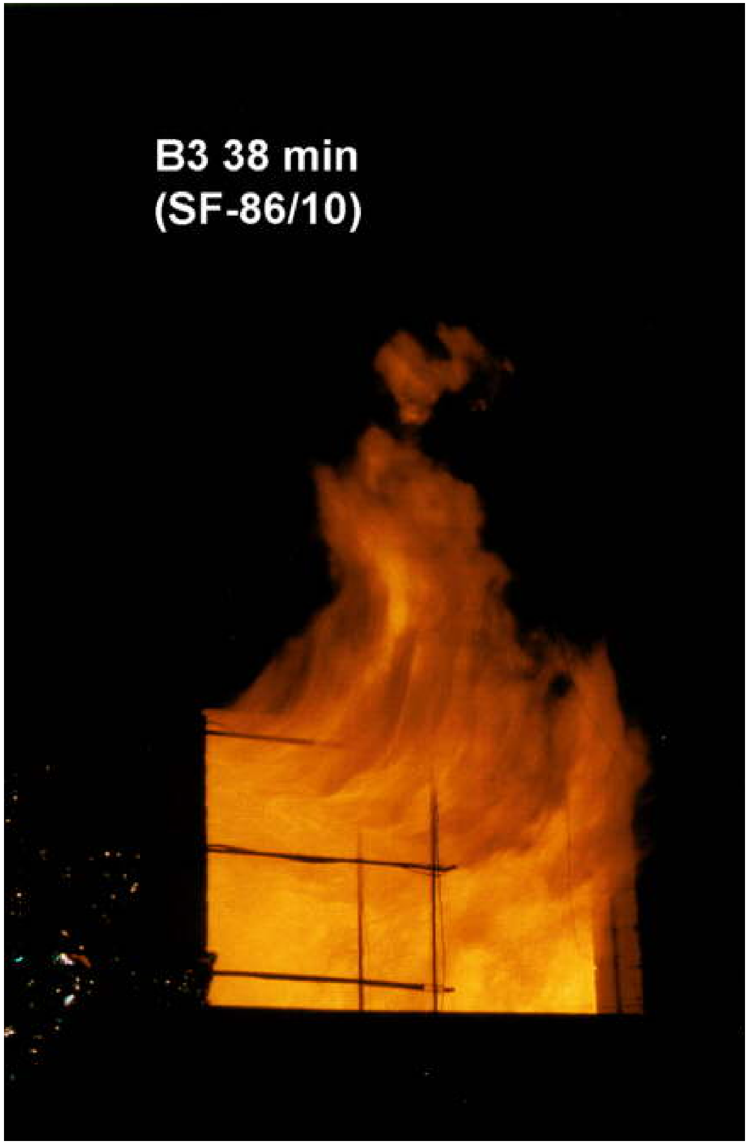




Fire simulations do not usually predict fire size so this was part of the input data

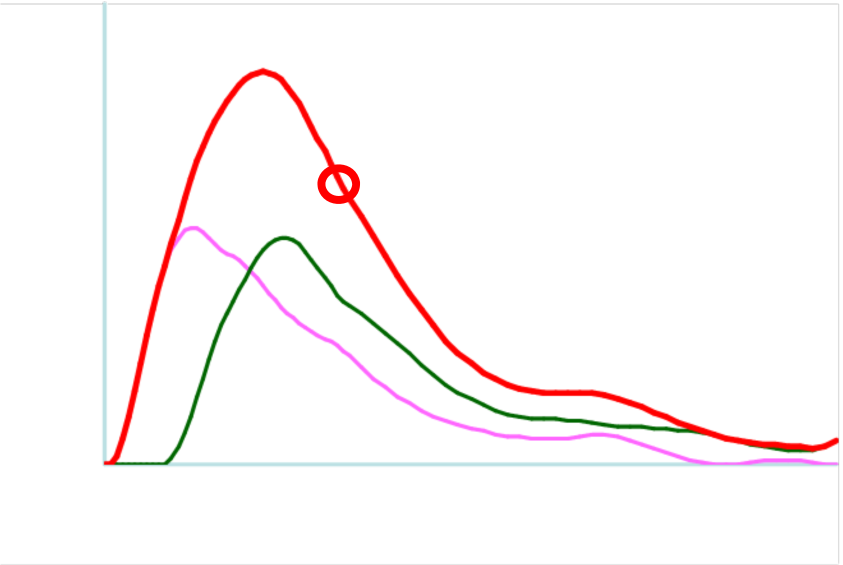
# Experiment: 8 minutes



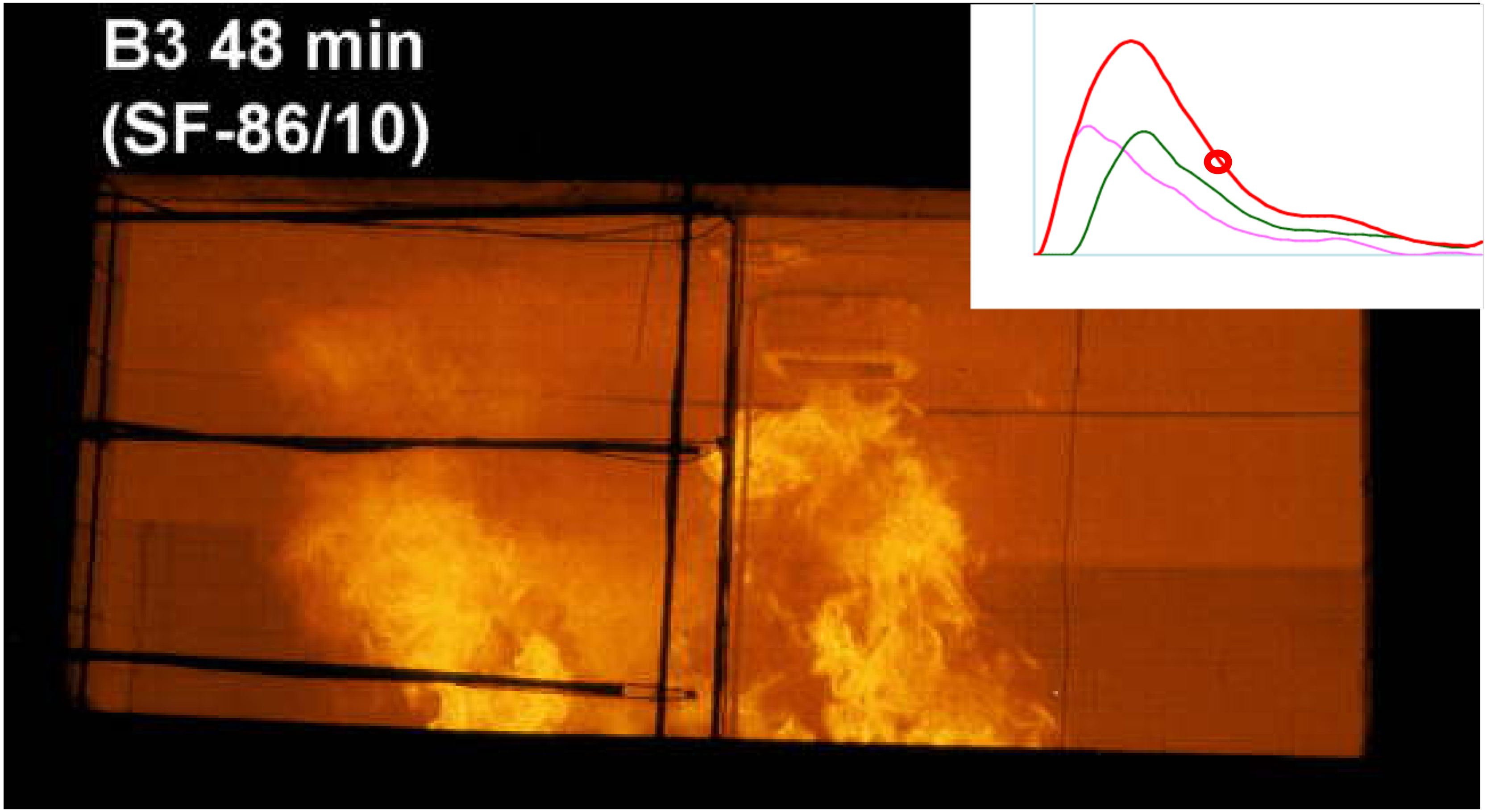
# Experiment: 38 minutes




- Room flashed over
- Flames emerging from window



# Experiment: 48 minutes

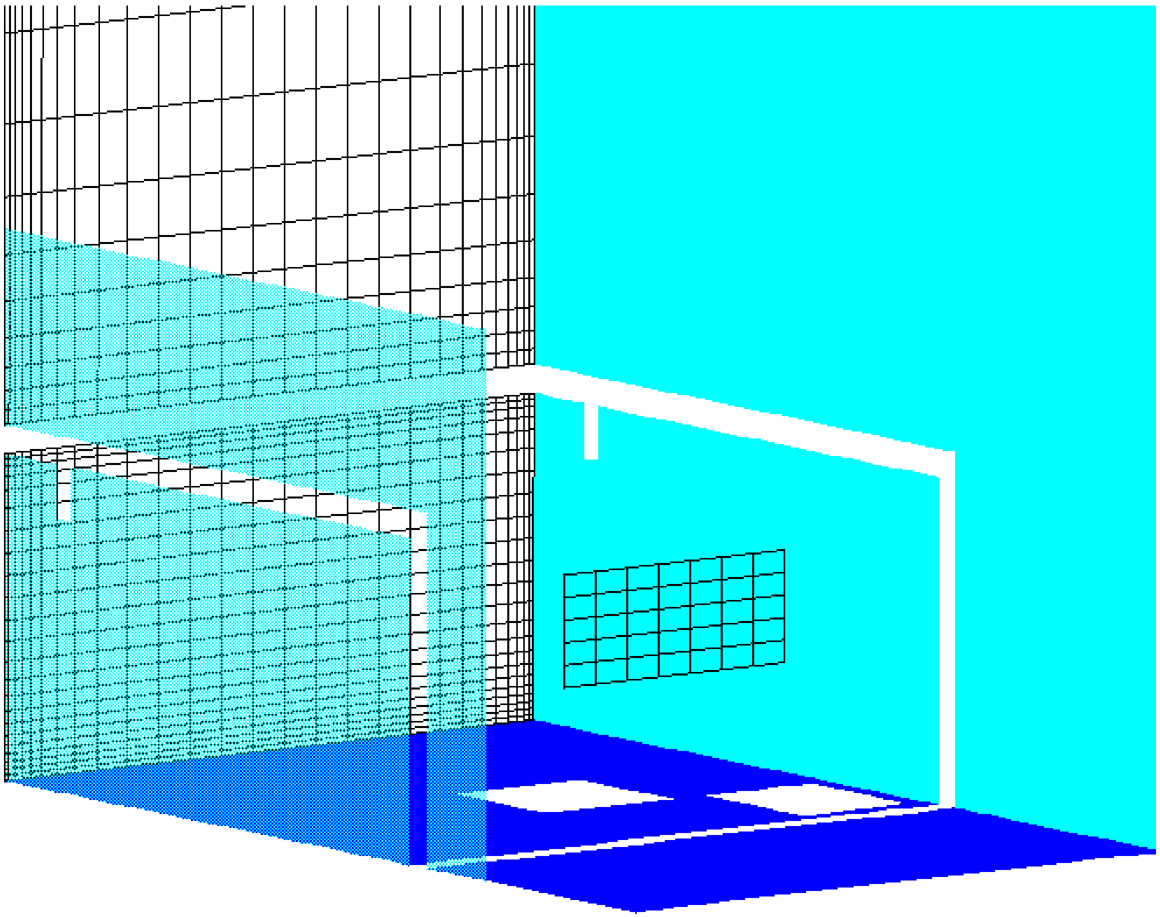


# JASMINE

- Finite volume CFD fire model
  - developed at FRS for more than 20 years
  - Based on early version of PHOENICS
  - validated for various smoke movement applications
- Six-flux radiation model
- Standard k-$\varepsilon$ turbulence model
  - with buoyancy modifications
- Specific heat & density
  - functions of species and temperature
- Solid surface temperature calculation
  - one-dimensional quasi-steady conduction approximation
- Two-second time-step
  - full two hour simulation

# Geometry and Mesh

- Domain extended into Test Hall

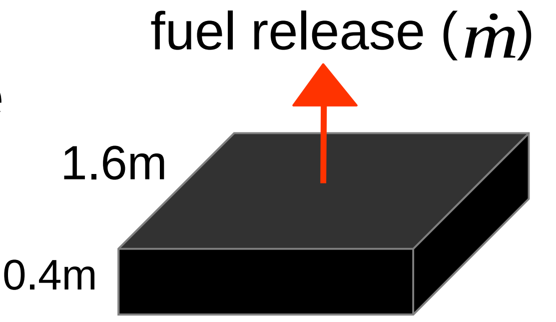


- 46,000 cells
  - finer grid at solid boundaries
  - grid sensitivity study with 370,000 cells

# Combustion Model

- Simplified crib
  - fuel released from top surface

$$\dot{Q} = \dot{m}\Delta H_{eff}$$



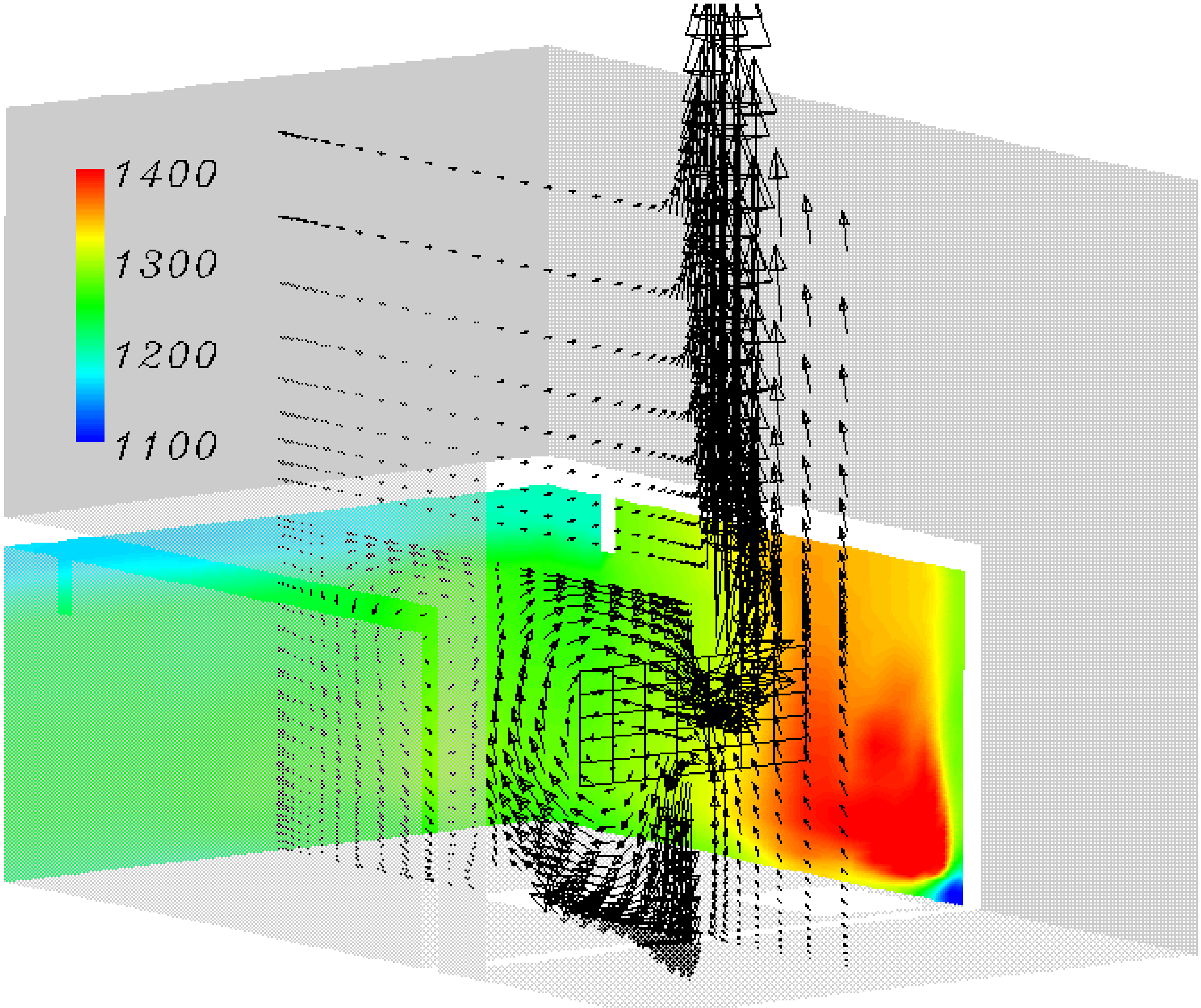

- Approximate one-step chemistry

$$CH_2O + O_2 \rightarrow CO_2 + H_2O$$

- Eddy dissipation reaction mechanism

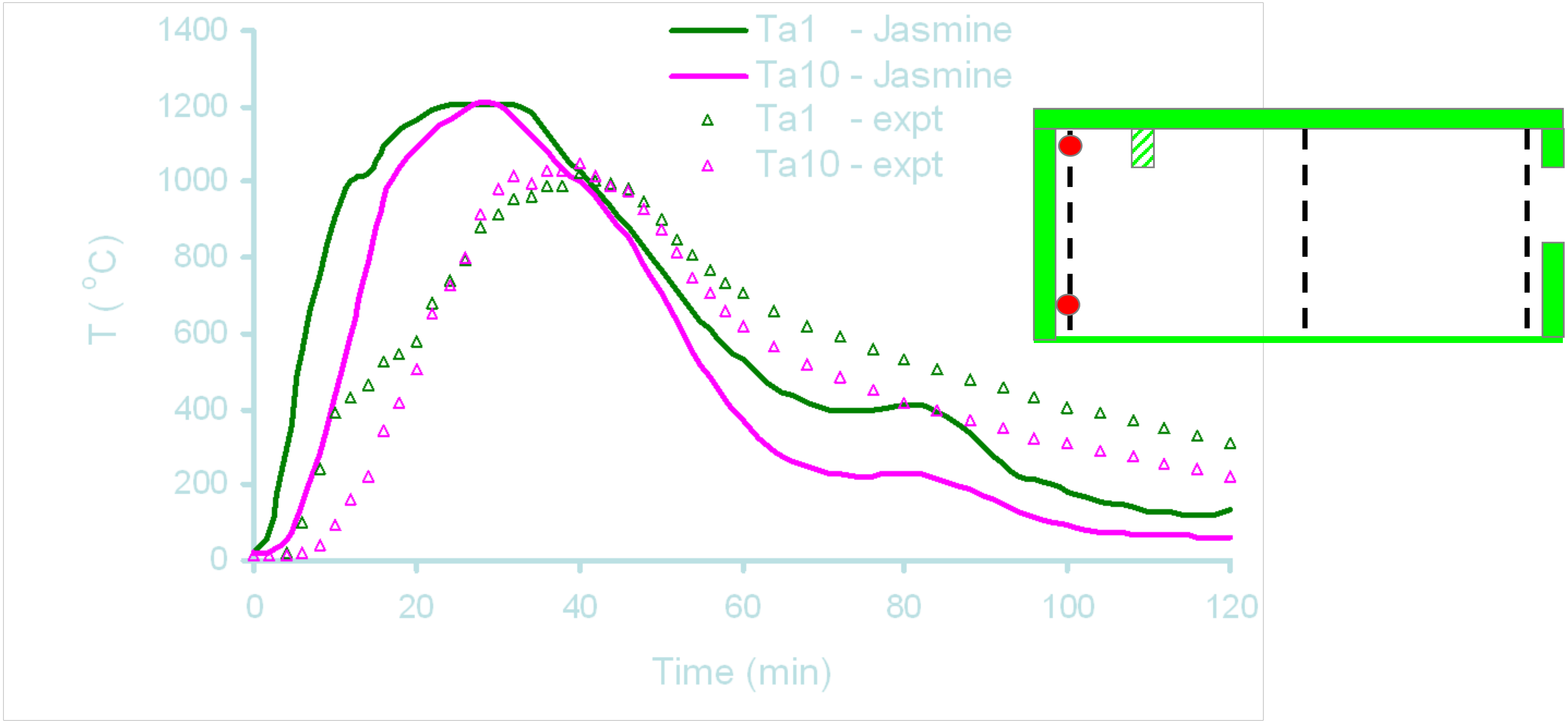
$$S_{fu} = -\rho \frac{\varepsilon}{k} C_R \min\left( m_{fu}, \frac{m_{o2}}{s} \right)$$

# Predicted temperature at flashover



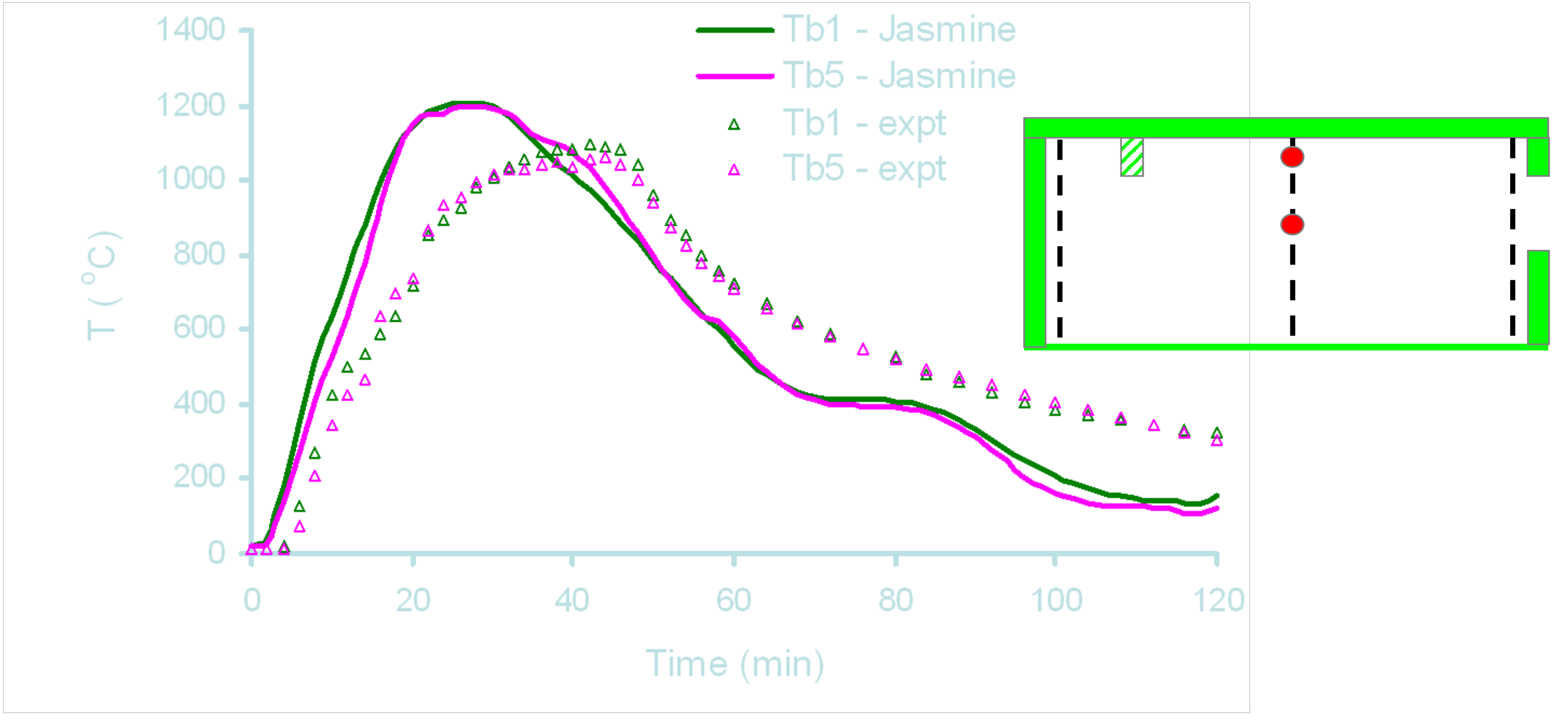
# Predicted & Measured Temperature

- Rear thermocouple tree



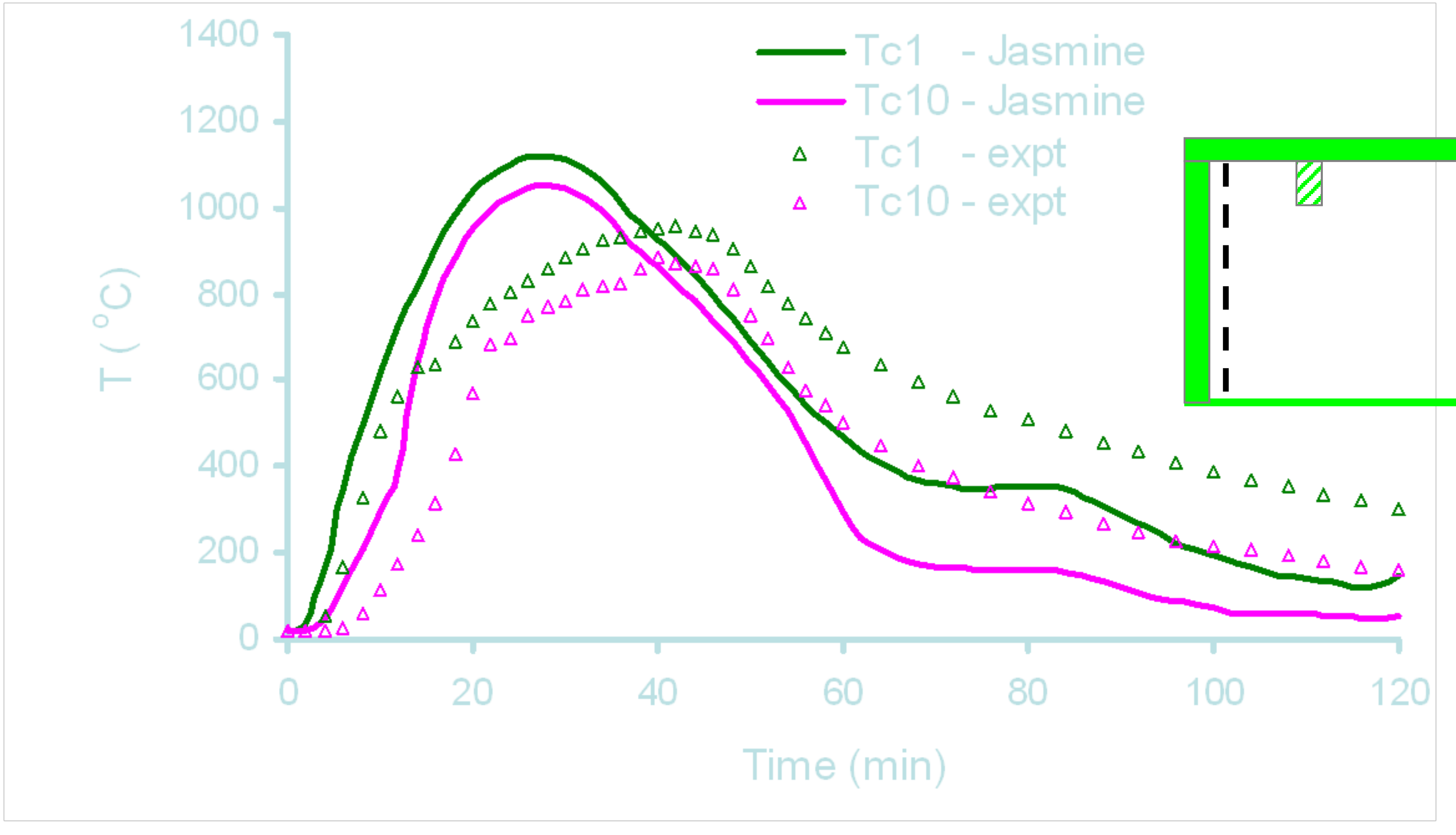
# Predicted & Measured Temperature

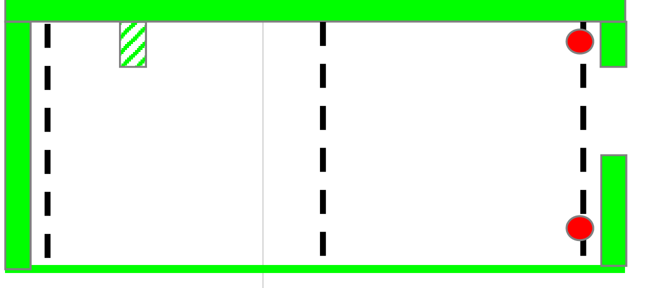
- Centre thermocouple tree



# Predicted & Measured Temperature

- Corner thermocouple tree



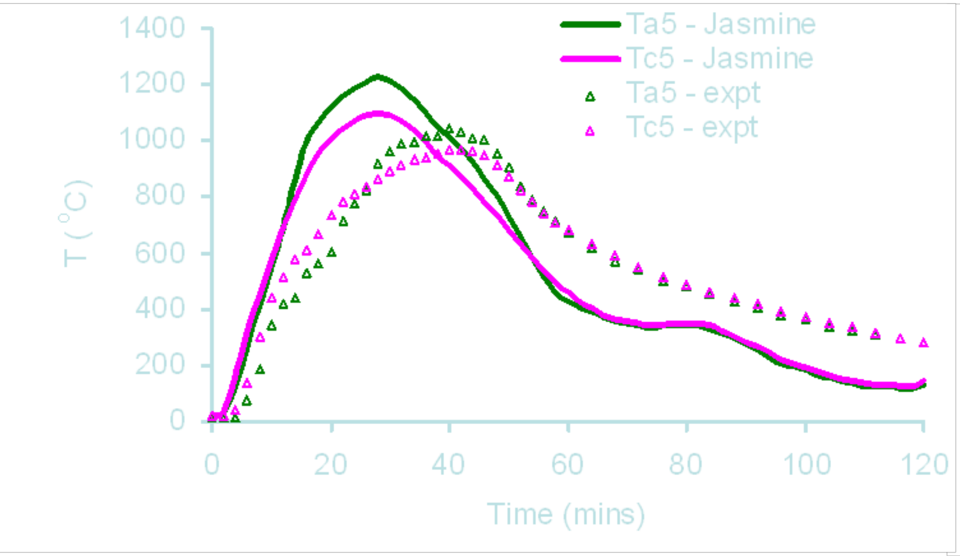



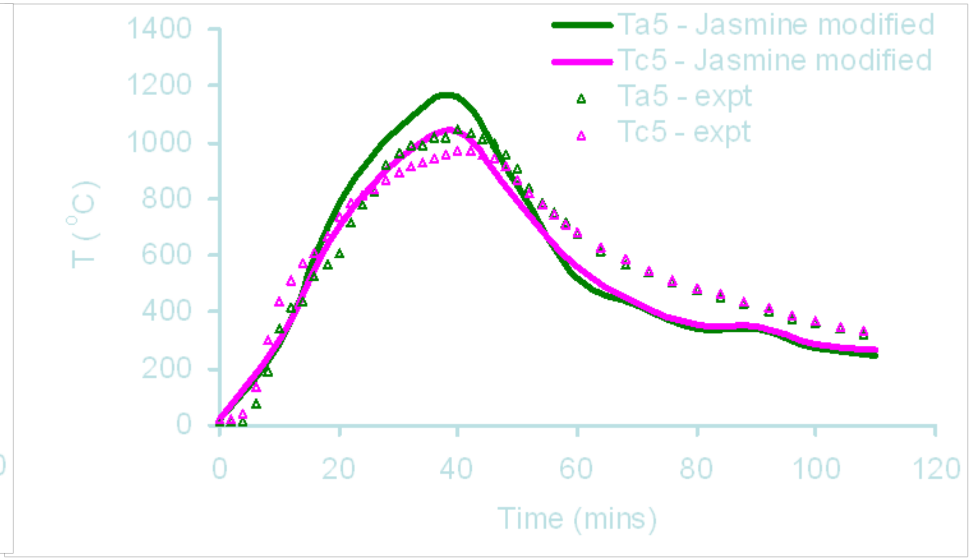
# Effective Heat of Combustion

- Constant value used for simulation

# Adjusted Temperature Prediction

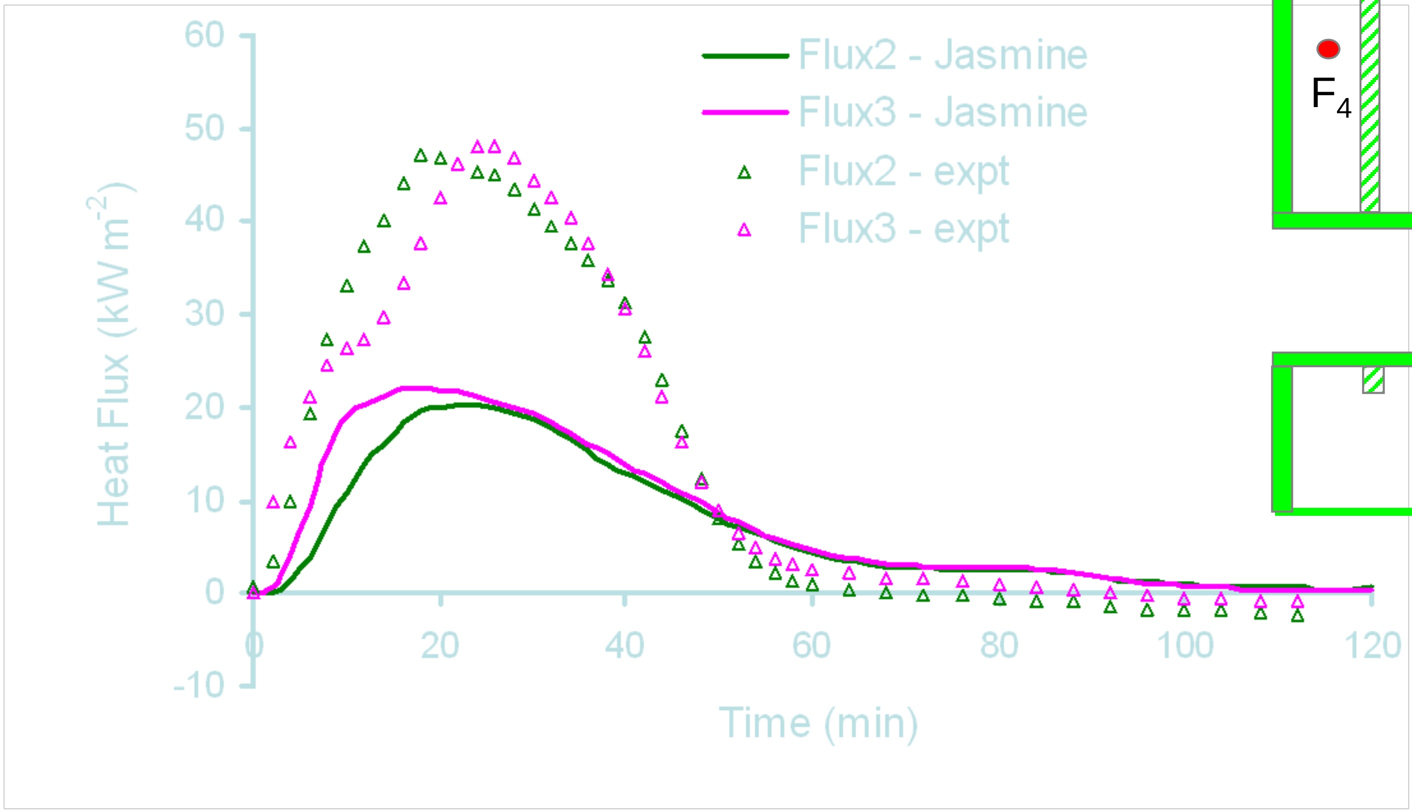
- Prediction 'modified' according to varying heat of combustion

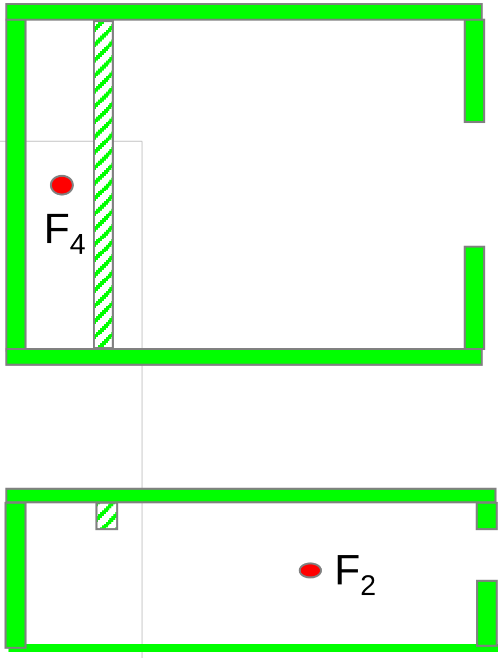


# Predicted & Measured Fluxes

- Conduction fluxes into ceiling and side wall

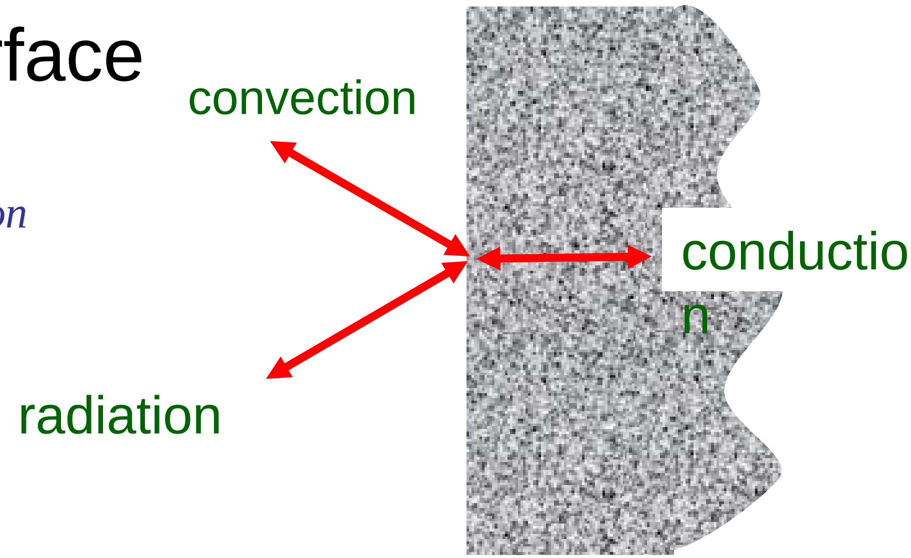


# Conduction Model

- Flux balance at surface

$$\dot{q}''_{conv} + \dot{q}''_{rad} = \dot{q}''_{conduction}$$




- One-dimensional quasi-steady conduction approximation

$$\dot{q}''_{conduction} \approx k\frac{\left(T_w - T_0\right)}{\delta}$$

$$\delta \approx 2\left(\frac{k}{\rho c}t\right)^{1/2}$$

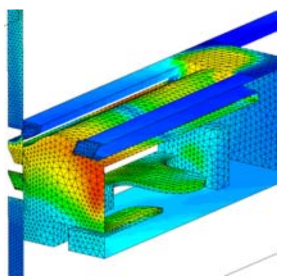
# Outcome of comparison

- Overall agreement between prediction and measurement good
  - peak temperatures within 15%
  - species concentrations similar
- Temporal shift and discrepancy in decay stage
  - variation in $\Delta H_{eff}$ an important factor here
- Solid boundary heat fluxes under-predicted during ‘flashover’
  - ‘simple’ quasi-steady conduction model
  - soot formation

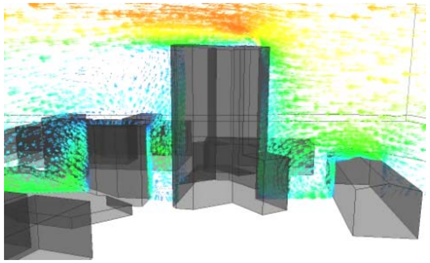
# Conclusions

- CFD has been demonstrated to accurately simulate a number of building related problems by comparison with measured data.

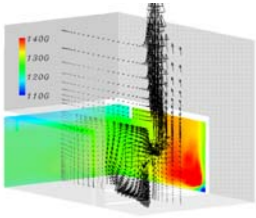
- BUT if it goes wrong…



**Ventilation**
Open a window,
loose energy



**Wind**
Discomfort
injuries



**Fire**
Large financial losses
People die

# Conclusions

- Fire and Low Energy Technologies
  - Better insulation (not just U value)
  - Better air tightness
  - Chilled ceilings
  - Phase change materials

- Some CFD issues
  - Free software(e.g. OpenFoam, FDS)
  - Training
  - Data sources
    - *Garbage in = Garbage out*

# Acknowledgements


- Colleagues from BRE
  - *Stewart Miles*
  - *Philippa Westbury*
  - *Geoff Cox*
  - *Suresh Kumar*