BASIC CONCEPTS FOR NATURAL VENTILATION OF BUILDINGS

Prof. Hazim Awbi

Technologies for Sustainable Built Environments Centre
University of Reading, UK

http://www.reading.ac.uk/tsbe

Email: h.b.awbi@reading.ac.uk

Natural Ventilation

In *Natural Ventilation* the airflow is due to wind and buoyancy through cracks in the building envelope or purposely installed openings.

Single-Sided Ventilation:

Limited to zones close to the openings

Cross-Ventilation:

Two or more openings on opposite walls - covers a larger zone than the single-sided openings

Stack Ventilation:

Buoyancy-driven gives larger flows

Windcacthers:

Wind and buoyancy driven - effective in warm and temperate climates

Solar-Induced Ventilation:

using the sun to heat building elements to increase buoyancy - more effective in warm climates

Hybrid Ventilation

In *Hybrid* (*Mixed Mode*) *Ventilation* the airflow is due to wind and buoyancy through purposely installed openings in the building envelope supplemented, when necessary, by mechanical systems.

The mechanical component of the hybrid system can be a fan for increasing the ventilation rate, and/or a heat exchanger for heating or cooling the outdoor supply air.

Airflow in Natural Ventilation

Factors Influencing the airflow through openings

- Wind speed
- Wind pressure
- Buoyancy (stack) pressure
- Characteristics of openings (C_d)
- Effective area of multiple openings

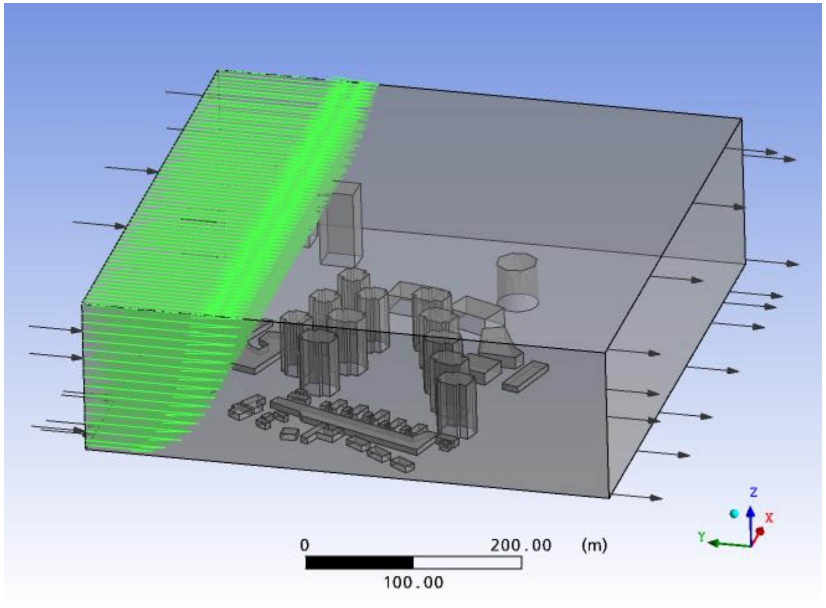
Indoor Environmental considerations

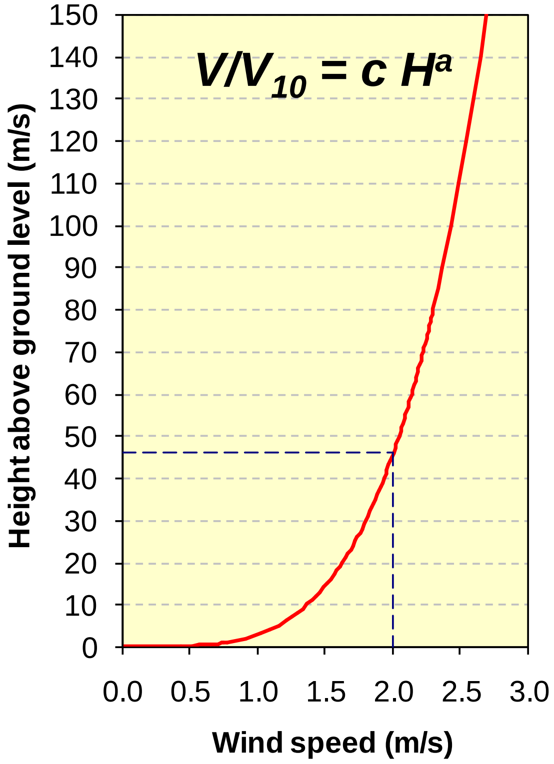
- Thermal comfort
- Indoor air quality

Wind Speed

Terrain factors for equation (2).

Terrain	*c*	*a*
Open flat country	0.68	0.17
Country with scattered wind breaks	0.52	0.20
Urban	0.35	0.25
City	0.21	0.33





Wind and Stack (Buoyancy) Pressures

Wind Pressure

$$P_w = ½ \rho V^2 C_p \quad (1)$$

The wind speed V is usually calculated at the height of the opening or a reference point on the building (e.g. roof), C_p is the pressure coefficient and ρ is the air density.

The expression $V/V_{10} = c\, H^a$ is commonly used. (2)

where V is the required wind speed at height H (m)
V_{10} is reference wind speed (at 10 m in open country)
c and a are the terrain factors depending on sheltering

Stack Pressure

$$P_s = - \rho\, g\, H\, (T_i – T_o/T_i) = - \rho\, g\, H\, (\Delta T/T_i) \quad (3)$$

where T_o is the outdoor air temperature (K)
T_i is the indoor air temperature (K)
H is height between two openings

Wind Pressure

The static pressure coefficient *(C_p)* is defined as:

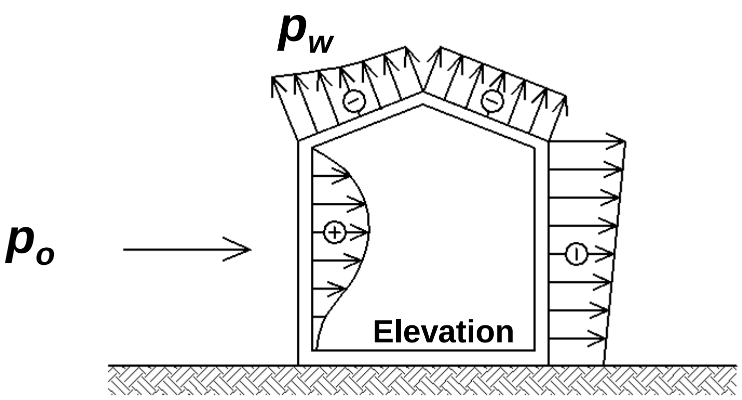
$$C_p = (p_w - p_o) / (½ \rho V^2) \qquad (4)$$

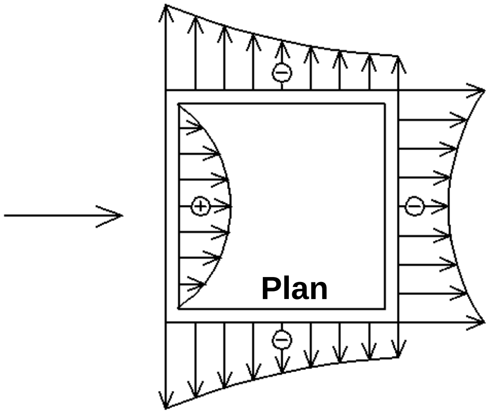
where p_w **= static pressure at some point on the building (Pa)**

p_o **= static pressure of the free stream (Pa)**

ρ **= density of free stream (kg/m³)**

V **= free stream velocity normally calculated at building height or other reference height (m/s)**





Wind pressure coefficients on a pitched roof building

Wind Pressure / Surface Coefficients (C_p) / …

The relative volume airflow rate error (r_i) for a specific pair of openings (*i*) can be defined as (Co'stola *et al, 2010*):

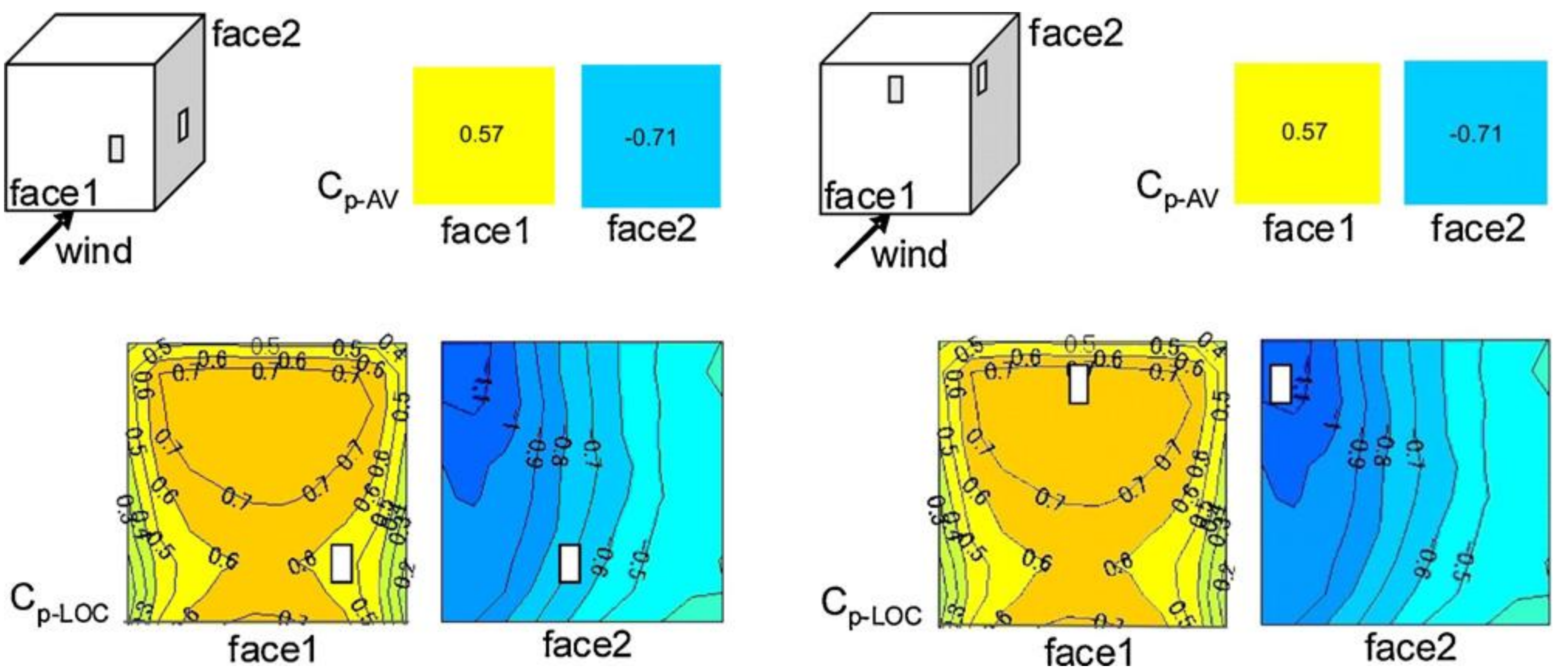
$$r_i = (V_{LOC} - V_{AV}) / V_{AV} = V_{LOC} / V_{AV} - 1$$

where V_{LOC} = air flow rate calculated using $C_{p\text{-}LOC}$
V_{AV} = air flow rate calculated using $C_{p\text{-}AV}$

$C_{p\text{-}LOC}$ = local pressure coefficient on a surface

$C_{p\text{-}AV}$ = average pressure coefficient over a surface

Wind Pressure / Surface Coefficients (C_p) / …



$C_{p\text{-}LOC}$ has the <u>same</u> value as $C_{p\text{-}AV}$ for the surface

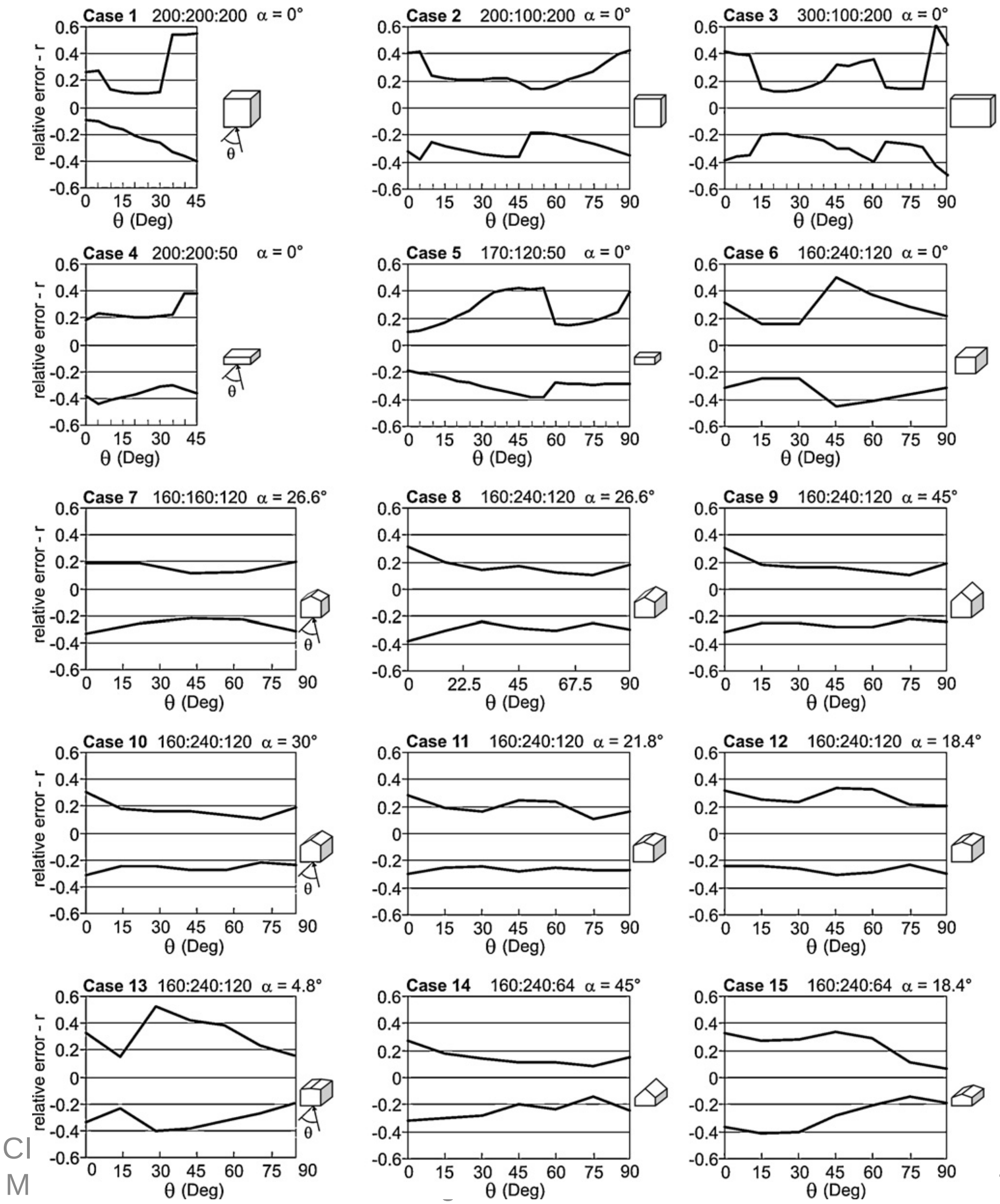
$C_{p\text{-}LOC}$ has a <u>different</u> value to $C_{p\text{-}AV}$ for the surface

(Co'stola *et al, 2010)*

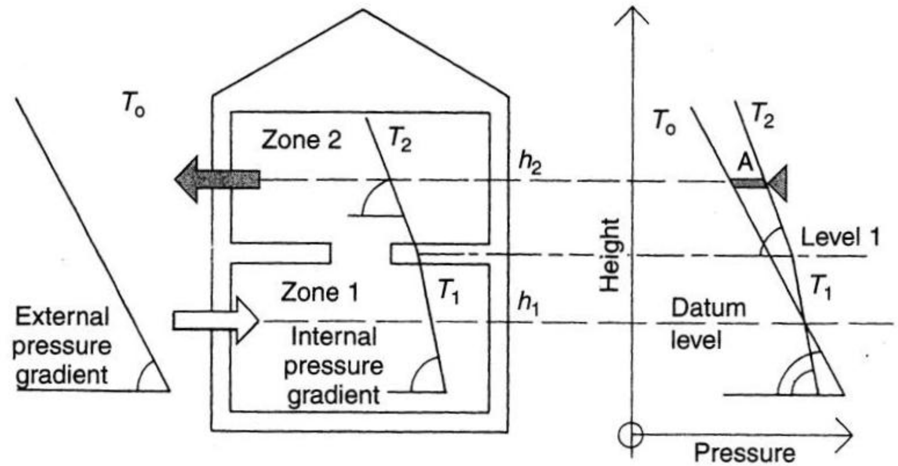
Wind Pressure / Surface Coefficients (C_p) / …

Upper and lower bound values of (*r*) as a function of the wind attack angle for 95% *Confidence Interval* when only pairs with the largest $\Delta C_{p\text{-}AV}$ are taken into account. If all surfaces are considered instead of those with $\Delta C_{p\text{-}AV}$ = maximum then the difference could be much larger.

(*α* is the roof pitch, Co'stola *et al* 2010).



Stack (Buoyancy) Pressure



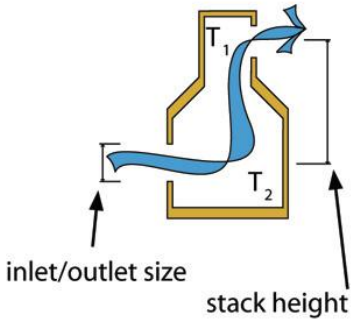
In this case the air enters the lower zone (Zone 1) from outside and leaves through an opening on the upper zone (Zone 2) with the flow passing through the opening between the two zones. In general, the temperature of zones 1 and 2 are not equal (here $T_2 > T_1$) and this results in different pressure gradients for each zone as shown in the Figure. The stack or buoyancy pressure for all the openings is calculated relative to that at the lowest opening. The pressure difference between the two external openings at h_1 and h_2 may be calculated using:

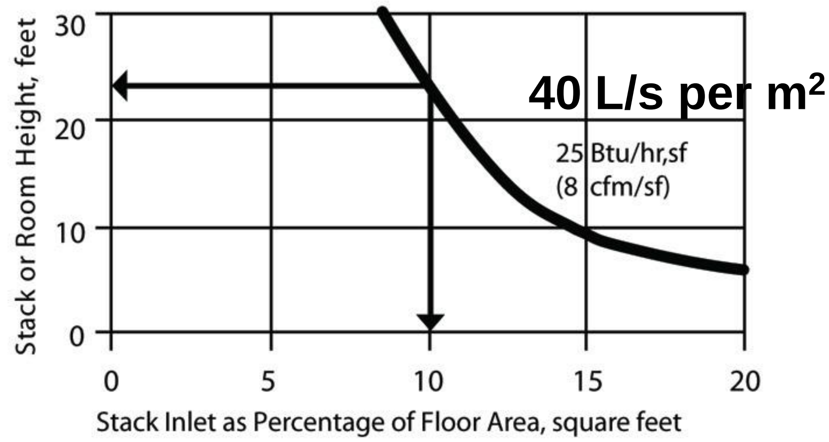
$$\Delta p = - \rho g [(z_1 - h_1)(1 - T_o/T_1) + (h_2 - z_1)(1 - T_o/T_2)] \quad (5)$$

where z_1 is the height of the first floor
T_2 & T_1 are the mean temperatures of zones 1 & 2.

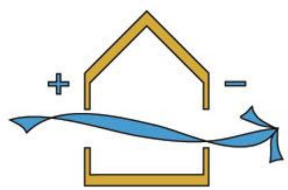
Simplified Flow Calculation

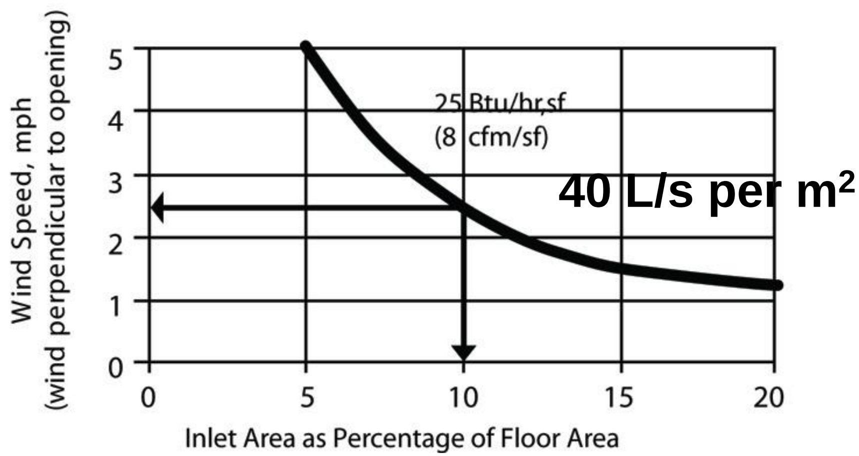
Stack





Cross-Flow





1 cfm/ft^2 = 5 L/s per m^2 1 mph = 0.45 m/s

(Brown & Huang, 2006)

Flow through an Opening

The flow through an opening can be calculated using the formula:

$$Q = A_{eff}\,[2\,\Delta p\,/\,\rho]^{1/2} \qquad (6)$$

where Δp is the pressure difference across the opening
ρ is the fluid density.

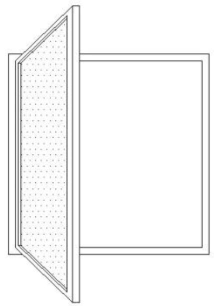
The effective area (A_{eff}) is given by: $A_{eff} = C_d\,A$ (7)

For natural ventilation openings the discharge coefficient (C_d) is not only dependent on type of opening and wind pressure but also on wind direction.

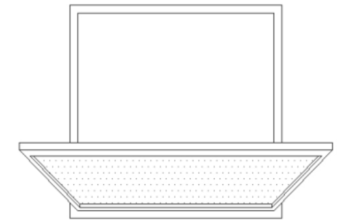
Data on C_d for natural ventilation openings is limited particularly its variation with wind direction. C_d is one of the main difficulties in manual calculation procedures and in network flow models for natural ventilation.

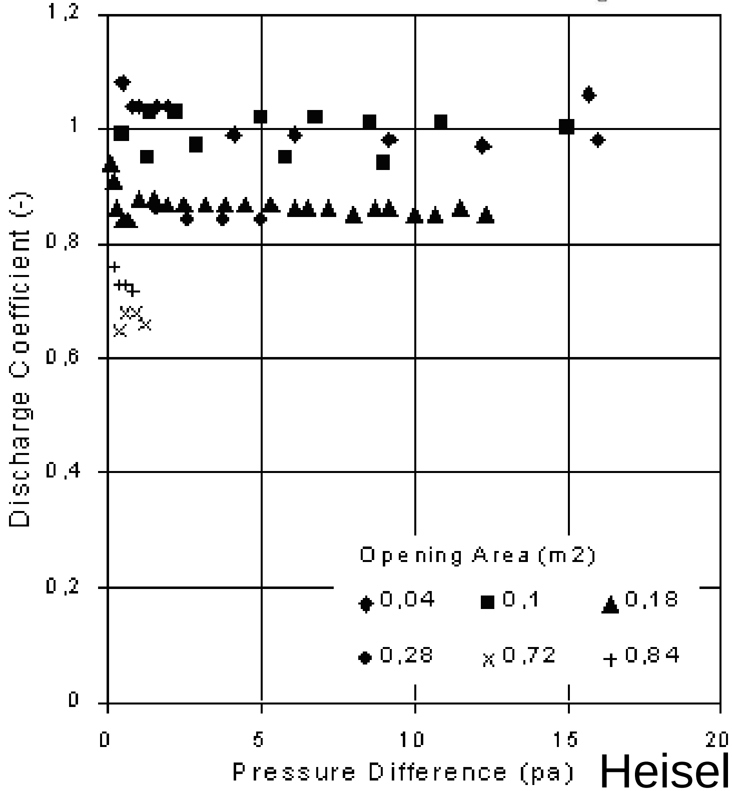
Flow through an Opening / …

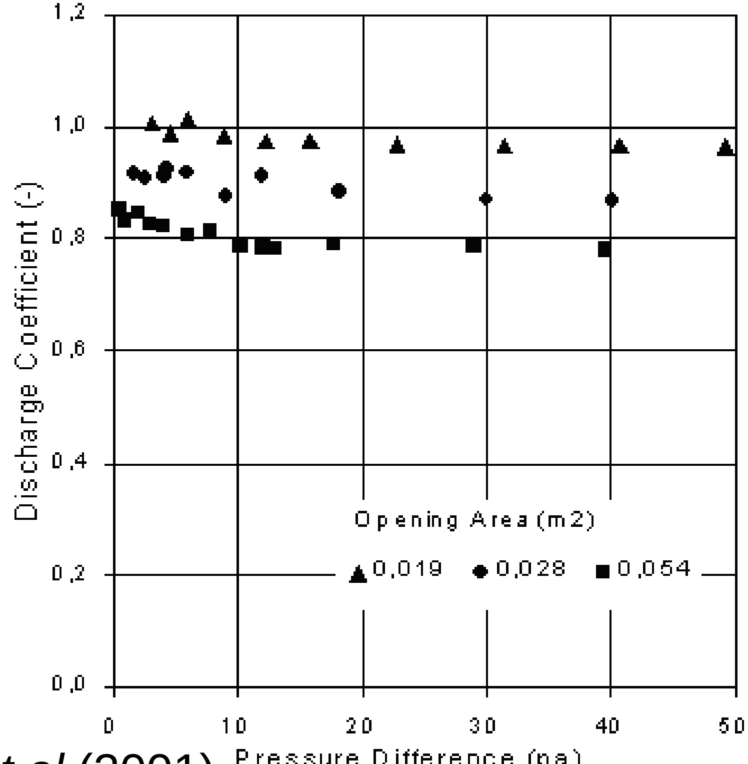
Discharge coefficient for side hung window



Discharge coefficient For bottom hung window







Heiselberg *et al* (2001)

Flow through a Large Opening

The flow through a large single opening can be calculated using the formula:

$$Q = C_d A \sqrt{(2 \Delta p / \rho)} \qquad (6)$$

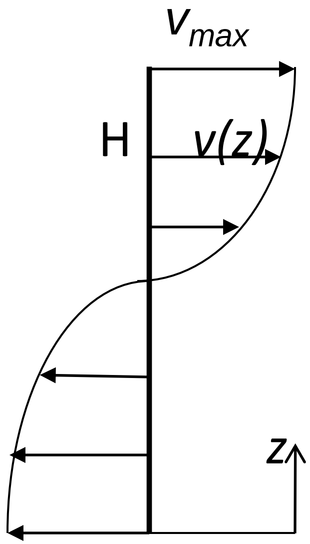
For buoyancy-driven flow through a small opening:

$$\Delta p = - \rho g H (\Delta T / T_i) \qquad (3)$$

$$Q = C_d A \sqrt{gH \frac{\Delta T}{T_i}} \qquad (9)$$

For buoyancy-driven flow through a large opening:

$$Q = \frac{C_d A}{3} \sqrt{gH \frac{\Delta T}{T_i}} \qquad (10)$$

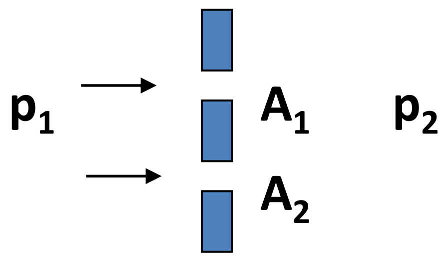


where *H* is the height of the opening. Since this airflow profile varies significantly along the opening height, the airflow is integrated over this height, producing the 1/3 constant in this equation (Awbi, 1996).

Effective area of multiple openings

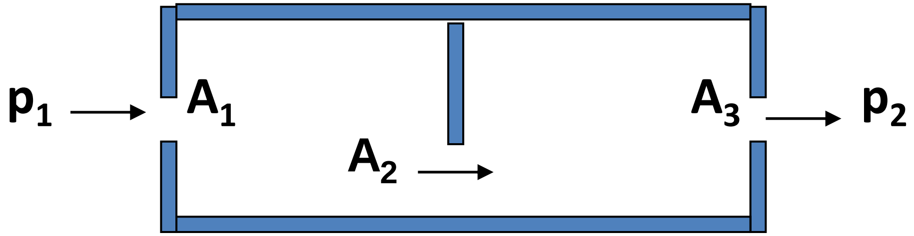
Openings in series:

$$A_{eff} = C_d A = C_{d1} A_1 + C_{d2} A_2 + \ldots$$



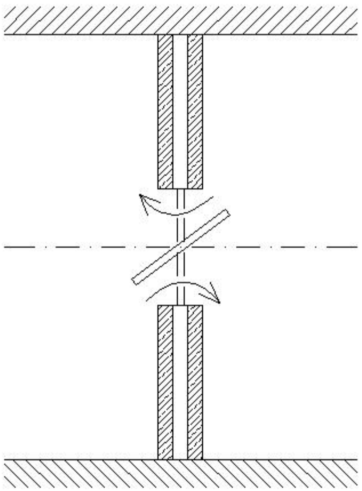
Openings in parallel:

$$1/A^2_{eff} = 1/(C_d A)^2 = 1/(C_{d1} A_1)^2 + 1/(C_{d2} A_2)^2 + \ldots$$

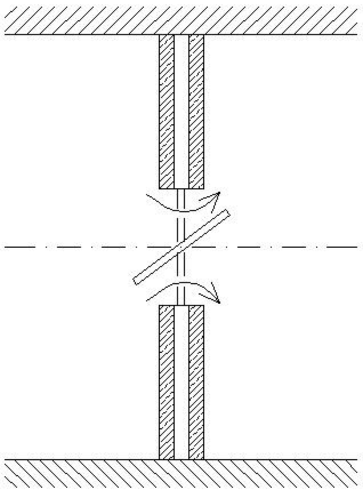


Flow through a centre pivoted window

What is the effective area?



Single-sided ventilation

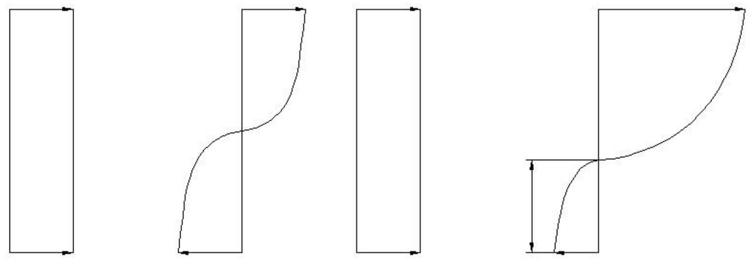


Cross ventilation

Combined Flow through an Opening

The total pressure due to wind, stack and a fan can be combined (as shown):

$$\Delta p_t = \Delta p_w + \Delta p_s + \Delta p_f$$



The total flow through an opening is: $Q_t \propto \Delta p_t^n$

where n **is a value between 0.5 and 1.0 depending on the opening dimensions and type of flow (laminar or turbulent), e.g.** $n = 0.5$ **for a large opening or 0.67 for a small opening.**

It is also possible to calculate the flow due to each pressure separately and combine these as follows:

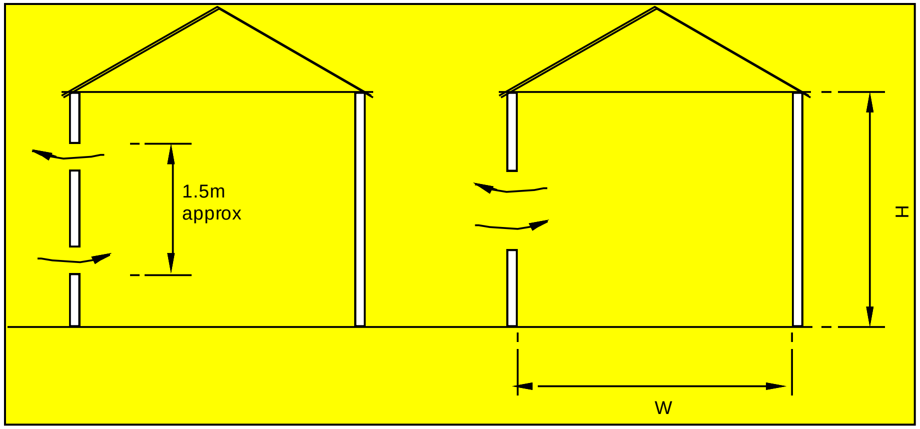
$$Q_t = [Q_w^{1/n} + Q_s^{1/n} + Q_f^{1/n}]^n \qquad (8)$$

A good approximation for large openings is as follows:

$$Q_t = [Q_w^2 + Q_s^2 + Q_f^2]^{1/2} \qquad (9)$$

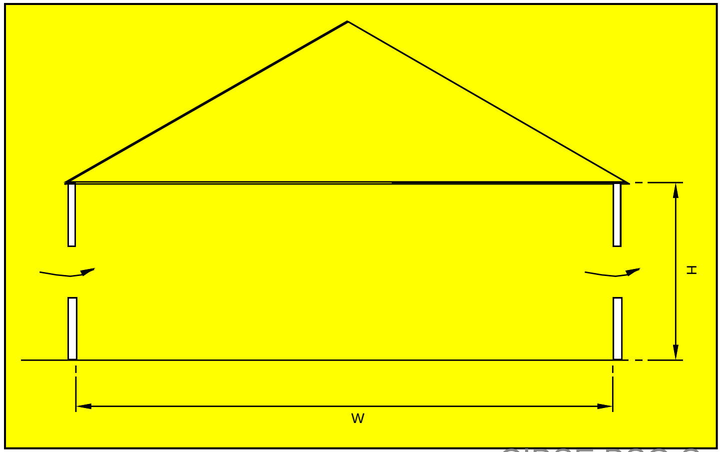
Natural Ventilation

Single-Sided Ventilation



The depth should be < 2.5 H

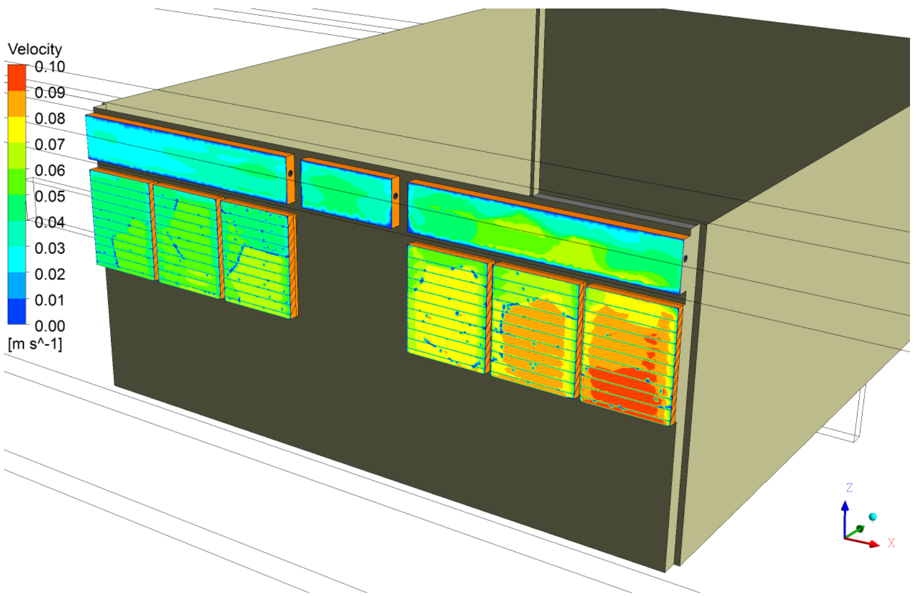
Cross-Ventilation

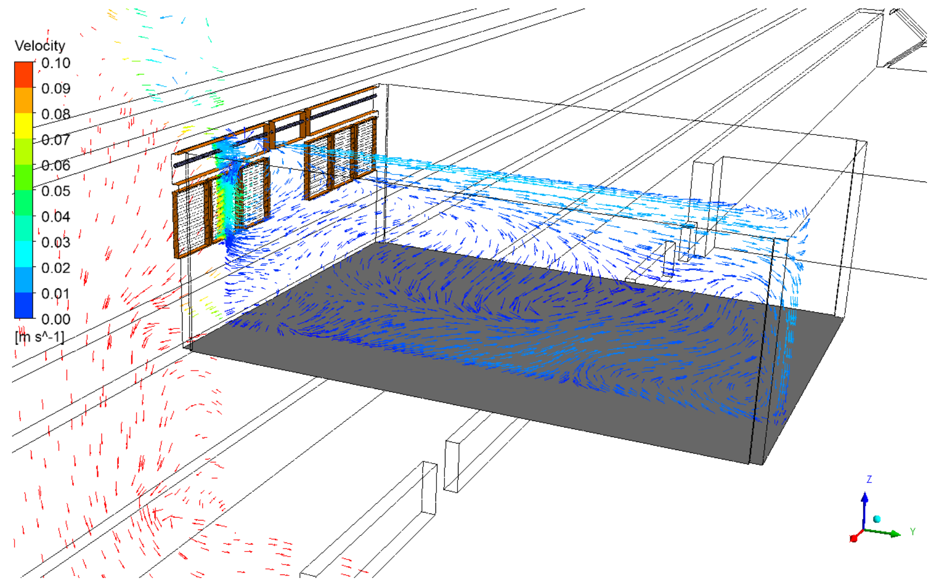


The depth should be < 5 H

Natural Ventilation/…

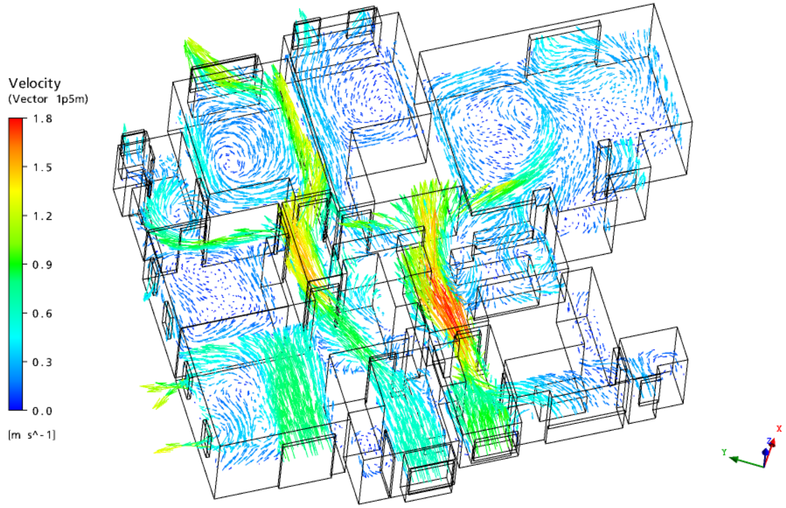
Single-Sided Ventilation – CFD Images

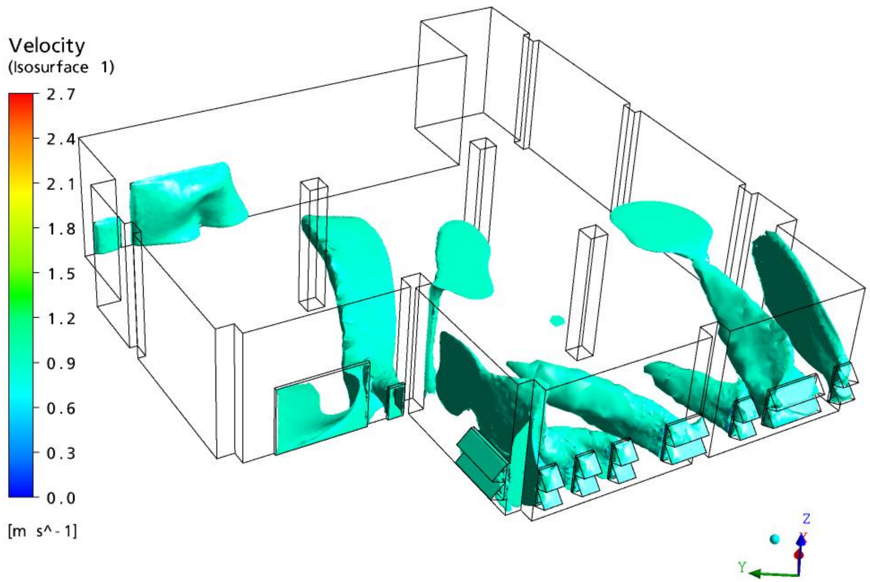




Natural Ventilation/…

Cross-Ventilation – CFD Images

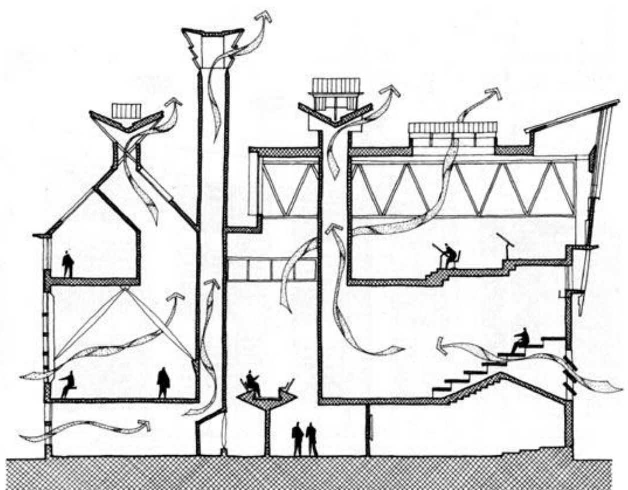




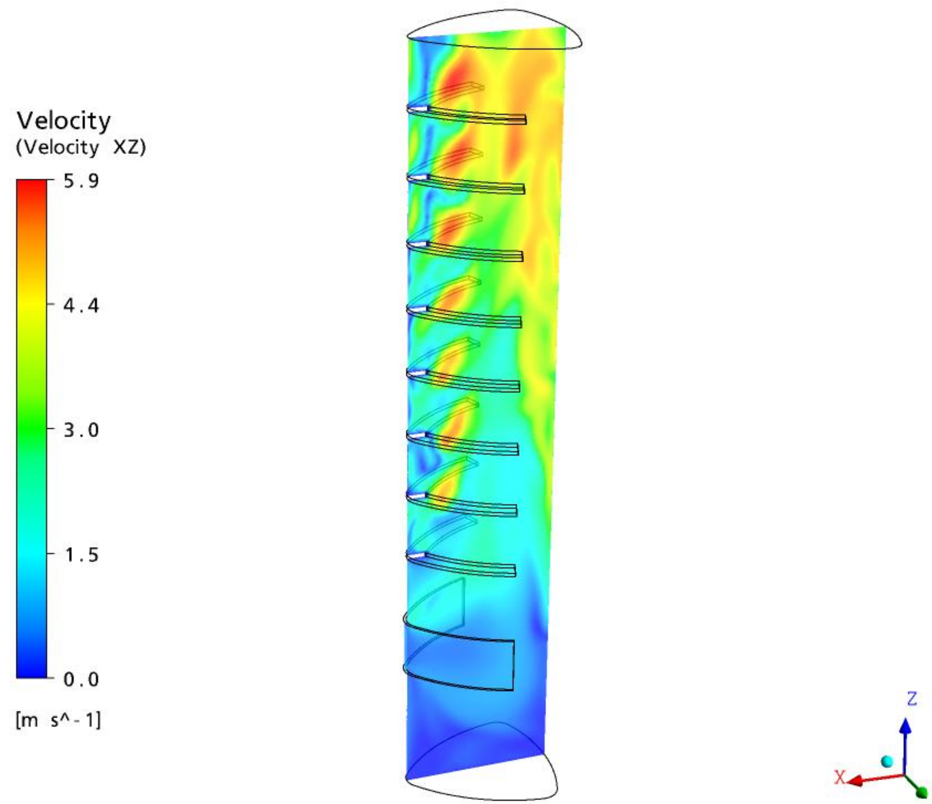
Natural Ventilation/…

Stack Ventilation

BRE Low Energy Office



Queen's Building (De Montfort University)

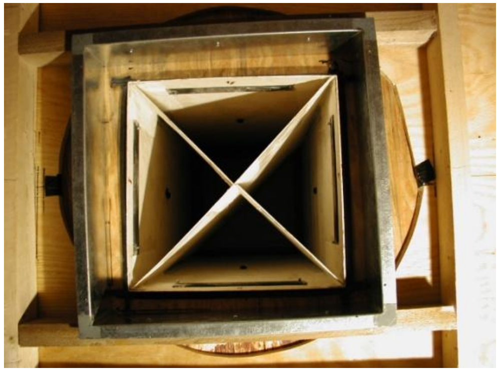


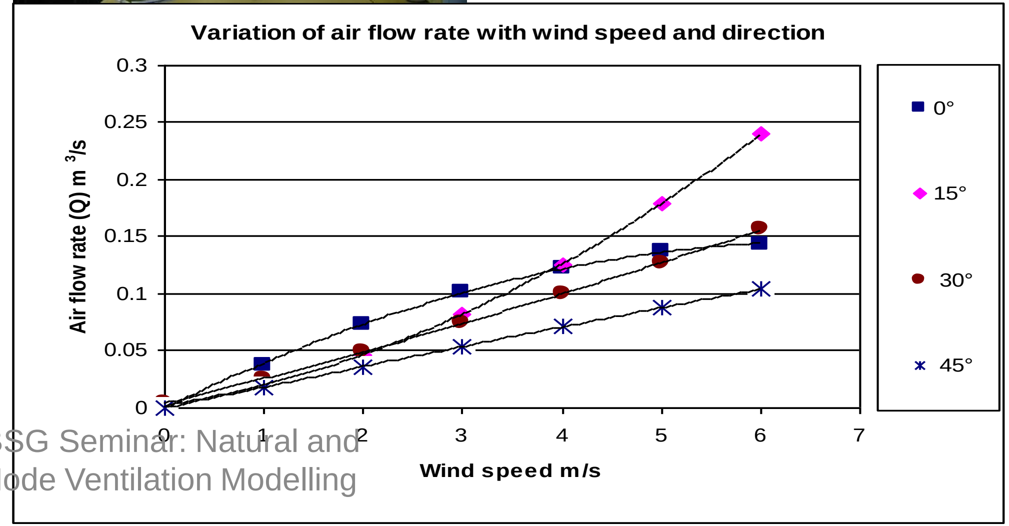
CFD simulation of FCU Exhausts in a Stack

Natural Ventilation/…

Windcathchers

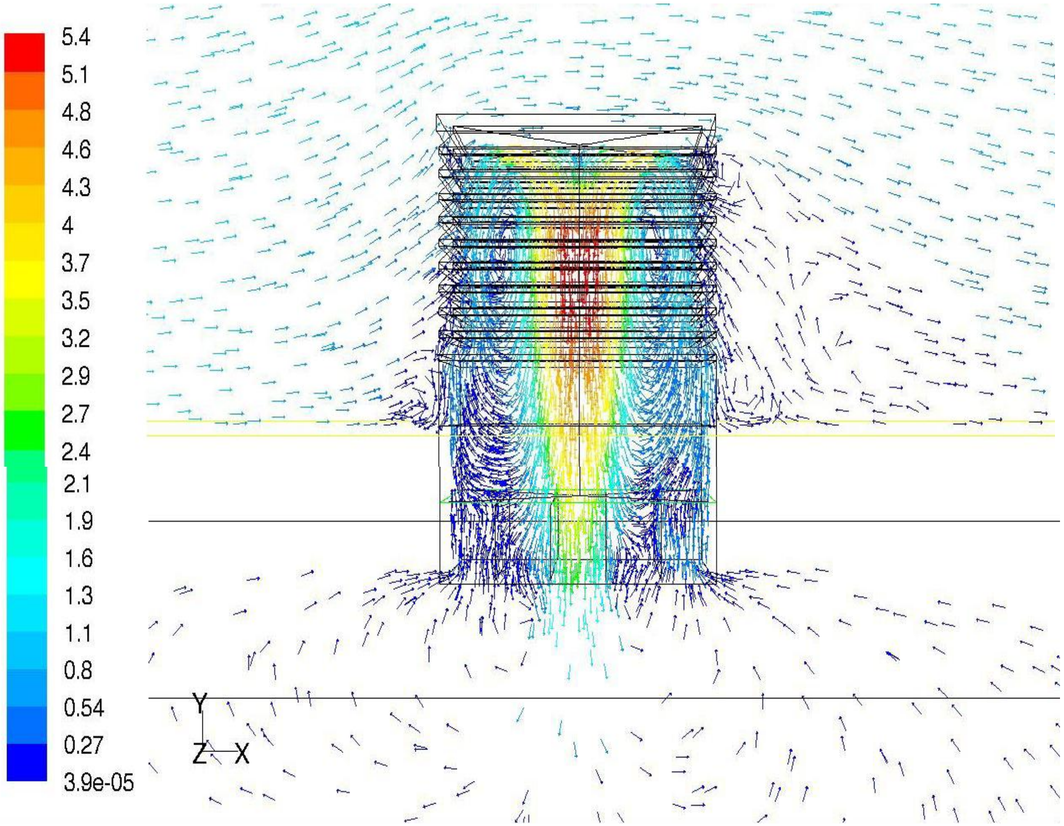
- Bi-directional flow with supply and extract
- Wind and buoyancy driven
- Flow-control either by the user or via sensors
- Suitable for large areas
- Usually requires roof access but flow can be ducted





Natural Ventilation/…

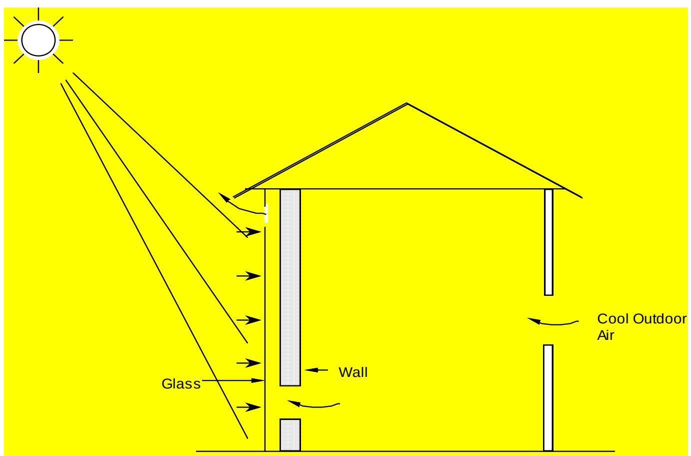
Windcathchers



Velocity vectors for the windcatcher with a fan blowing air centrally down (Hughes & Ghani, 2008)

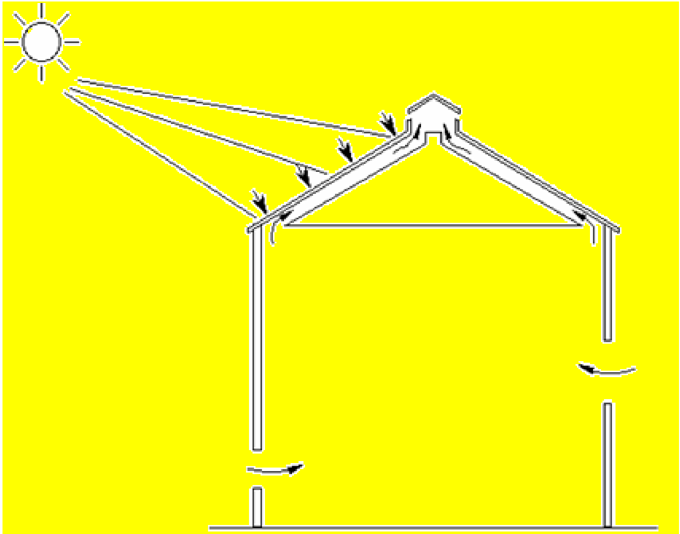
Natural Ventilation/…

Solar-Induced Ventilation



Solar Wall (Trombe Wall)

Solar Chimney

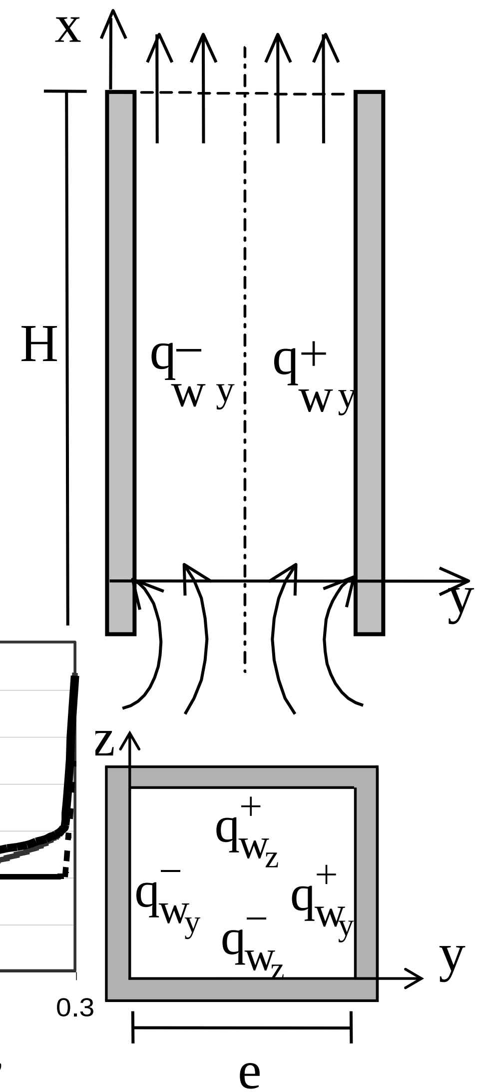


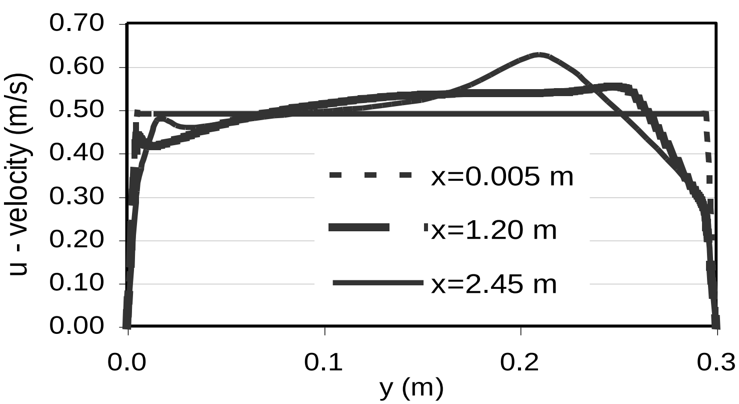
Solar Roof

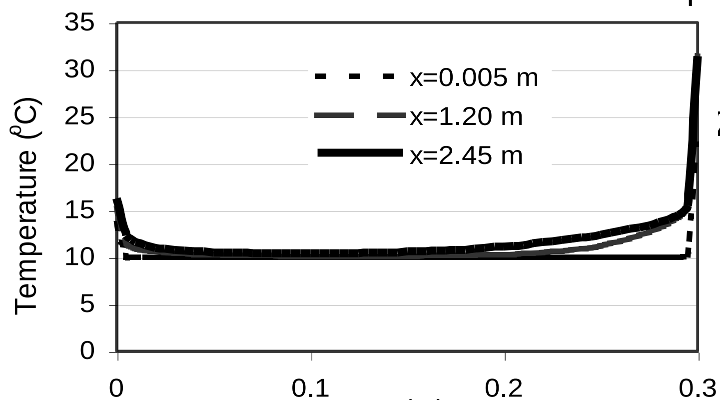
Natural Ventilation/…

Solar Wall

$$\dot{m} = \sqrt{\frac{\left(H - \frac{\ln(1+\Lambda H)}{\Lambda}\right)\rho_{s,0}^2 g}{\frac{2aH\,Re_H^n}{S^2 D_h}\left(\frac{1}{n+1}+\frac{\Lambda H}{n+2}\right)+\frac{1}{2}\left(\frac{1}{C_d S}\right)^2+\frac{\alpha \Lambda H}{S^2}}}$$



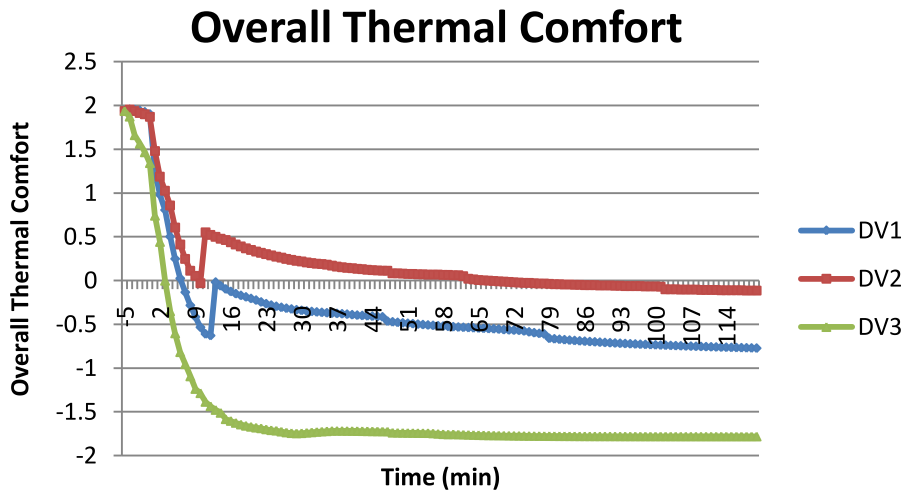




CFD prediction of temperature & velocity profiles in channel (Rodrigues, da Piedade & Awbi 2000)

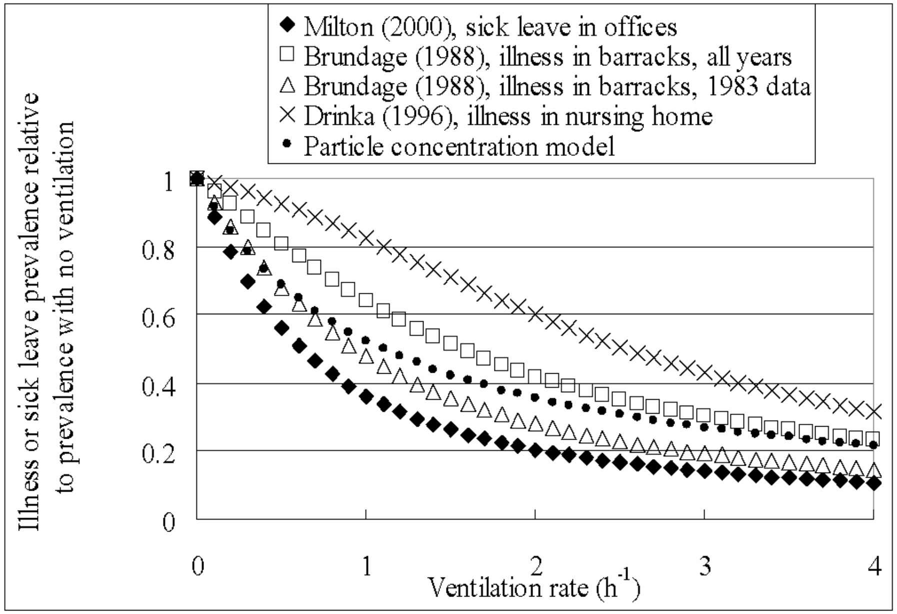
Thermal Comfort

- Standard thermal comfort models (e.g. Fanger's) are not usually suitable for naturally ventilated buildings as they are based on <u>uniform</u> thermal environment.
- Adaptive thermal comfort models are often used to allow for zone and diurnal/seasonal variations in the indoor thermal conditions.
- Thermal comfort models that are developed for non-uniform thermal environments are more suitable, e.g. Fiala's model and the CBE (Center for the Built Environment , University of California, Berkeley) model.



Indoor Air Quality

As control of ventilation rates is difficult for naturally ventilated buildings, attention should be given to the minimum ventilation rate that a system can be expected to provide. If this is below the minimum recommended rate for the building then a hybrid option should be contemplated. Failure to achieve this could cause poor indoor air quality issues and possible SBS symptoms.



(Fisk, et al. 2003)

CFD for Natural Ventilation

Why Use CFD for Natural Ventilation?

- **Lack of confidence in pressure coefficient data for wind-driven flow calculations**
- **Lack of knowledge of effective opening area (i.e. C_d)**
- **Difficulty in differentiating between air supply and extract openings (dependent on wind direction, turbulence, temperature difference, etc.)**
- **Calculations are needed for different wind directions but classical methods cannot accurately predict the effect of wind direction**
- **Usually room air movement predictions are needed in addition to air flow calculation**
- **The interaction between mechanical and natural systems is difficult to predict by standard calculation methods**

CFD Modelling of Natural Ventilation

❑ Building model

The translation of a real building into an "electronic model". Most CFD codes can import CAD files but this still requires some engineering and modelling skills.

❑ Air inlets and outlets

- **Air supply devices (grilles, louvers, etc.) can be complex and some simplification is needed.**
- **Understanding of device airflow characteristics will provide more accurate modelling.**

❑ Obstructions

Both internal and external flows are greatly influenced by the size and location of obstructions. Can be passive or emit heat and pollution.

CFD Modelling / …

❑ Heat and pollution sources

Internal or external heat and pollution sources should be accurately positioned and emission of heat and/or pollution from them should be specified.

❑ Computational grid

External flows usually require a large number of computational cells for accurate results. Therefore, convergence can sometimes be difficult.

Turbulence modelling can be challenging specially where large areas of flow separation occur.

❑ Simulation

Sometimes a dynamic simulation is necessary which can be very demanding.

CFD Modelling Techniques

There are two approaches –

Internal flow simulation:

This requires boundary conditions at inlets & outlets which may be obtained from wind pressure data for the openings or CFD simulations for the whole site.

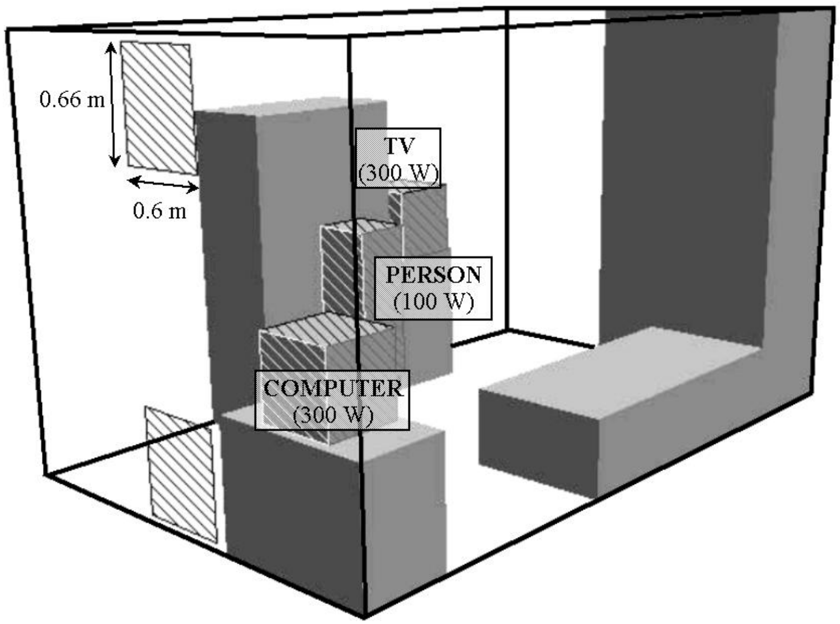
Coupled Internal and external flow simulation:

This requires climatic data, wind profiles and sheltering of surrounding structures.

Examples of CFD Modelling

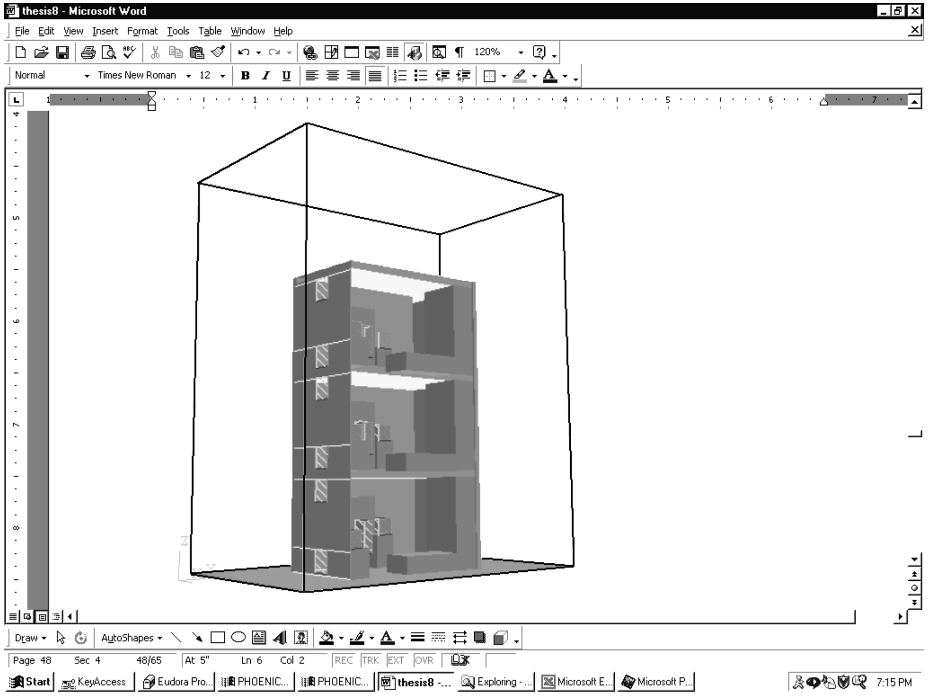
Single-Sided Ventilation (Buoyancy only)

Room with lower and upper openings



Room dimensions: 4.7 x 2.9 x 2.8 m
Room load: 700 W (50 W/m^2)

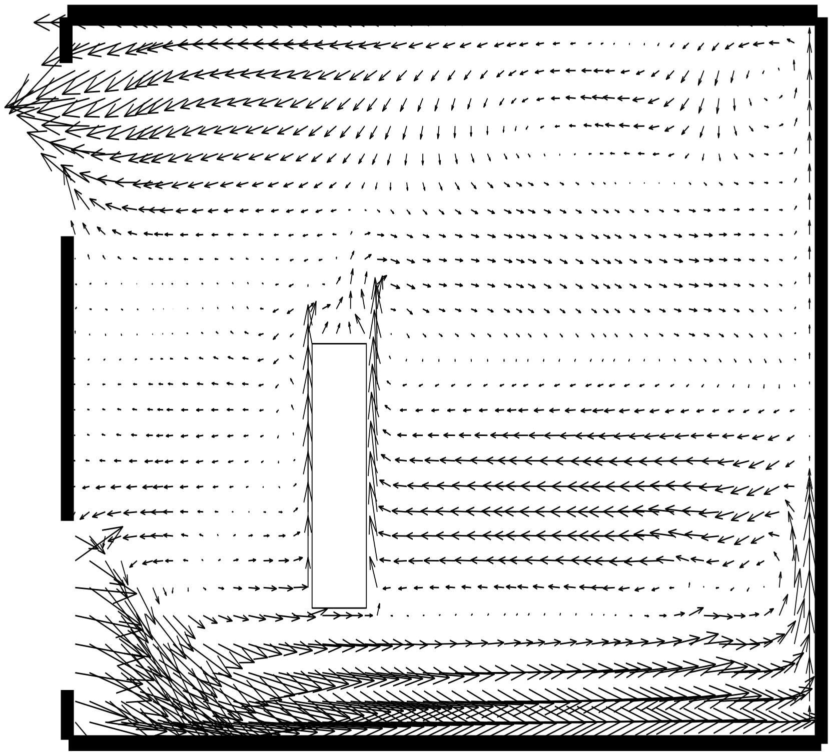
3-Storey Dormitory Building



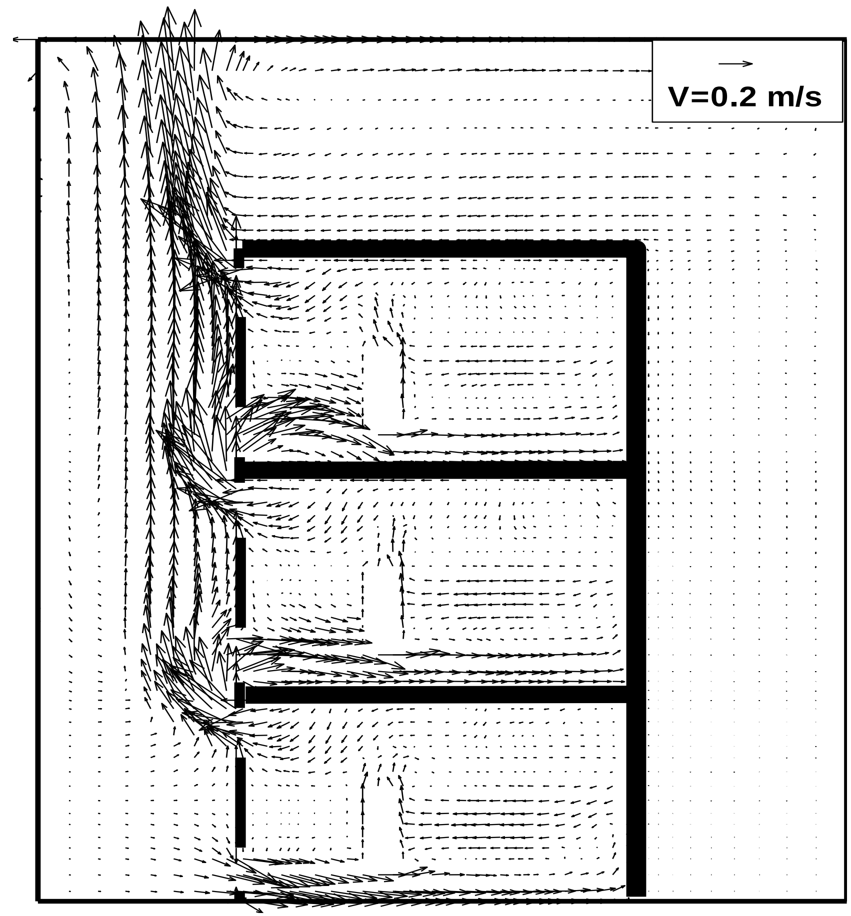
(Allocca, Chen & Glicksman, 2003)

Single-Sided Ventilation \ cont …

Buoyancy-Driven Ventilation



Internal flow modelling <u>only</u> with zero pressure at openings and zero gradients for velocity and temperature (Allocca, Chen & Glicksman, 2003).

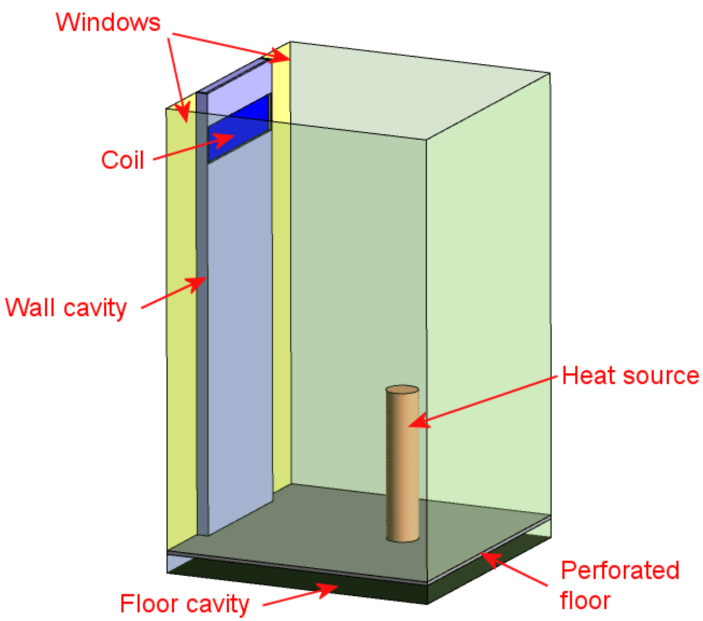


Internal <u>and</u> external flow modelling. The interaction between the flow in rooms at different level is clearly visible. This is due to the plume rising from low to high levels.

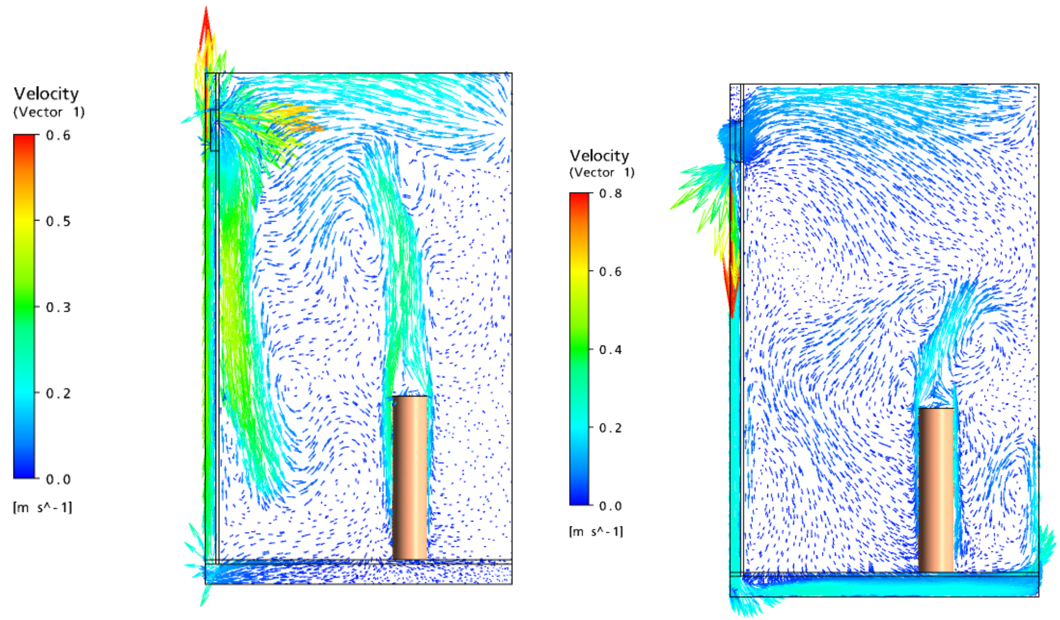
Examples of CFD Modelling / …

Passive Ventilation

Floor displacement flow by passive air ventilation: cooling coil, wall and floor cavities, perforated floor



Room model



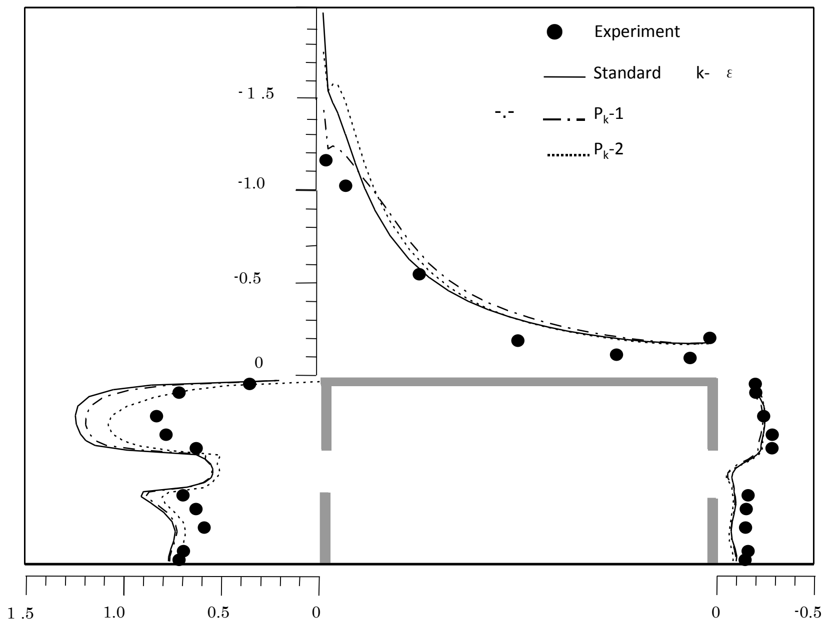
Cavity wall not insulated (40 W/m²) on wall

Cavity wall insulated

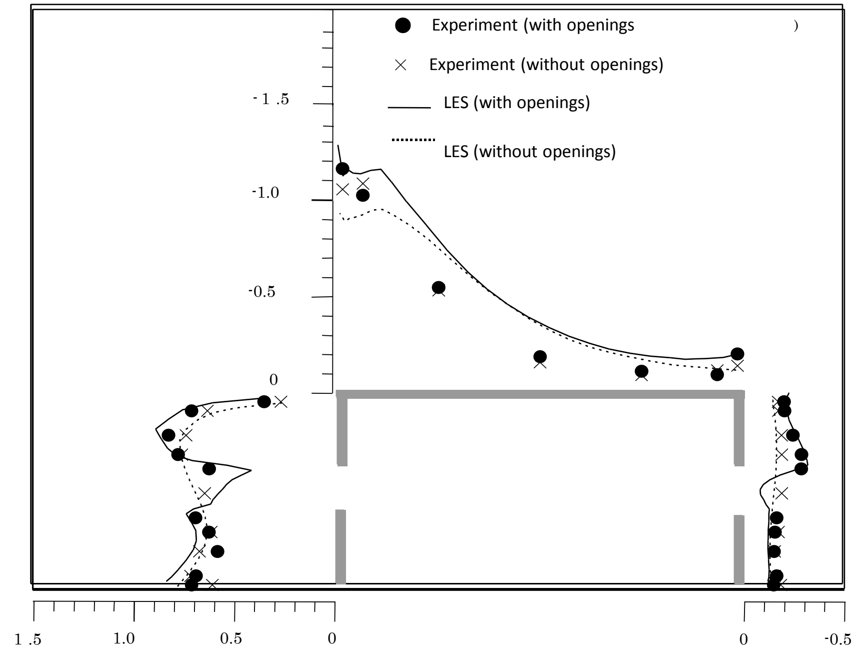
Velocity vectors in a vertical plane across the middle of the model perpendicular to the external cavity wall with cooling coil

Examples of CFD Modelling / …

Pressure Distribution



Pressure coefficients from standard *k-ϵ* turbulence model with wind-tunnel values



Pressure coefficients from LES turbulence model with wind-tunnel values

Distribution of pressure coefficients around the building (Kurabuchi, *et al*, 2000)

CONCLUSIONS

- **An understanding of the basic principles are necessary for meaningful modelling of naturally ventilated buildings.**
- **Attention should be given to representing opening geometry, pressure and other flow characteristics**
- **To be able to evaluate thermal comfort and indoor air quality CFD is often required.**
- **Where possible coupling of the external wind flow with internal flow should be used in a CFD simulation.**

Thank you

Questions?