COMPARISON OF ZONAL AND CFD MODELLING OF NATURAL VENTILATION IN A THERMALLY STRATIFIED BUILDING

Y. Li, A. Delsante, J. G. Symons and L. Chen

Advanced Thermo-fluids Technologies Laboratory
CSIRO Building, Construction and Engineering, Melbourne, AUSTR...

ABSTRACT

This paper compares two well-known modelling approaches for natural ventilation in a multi-zone building with thermal stratification and large openings. The zonal approach in this paper assumes a fully mixed condition in each zone, and considers the bi-directional flows through all large openings. The zonal model is integrated into a thermal analysis code to provide simultaneous prediction of both ventilation flow rates and air temperatures in each zone. The CFD approach uses a finite-volume method for turbulent flows. A simple pressure boundary condition is used at all external openings in CFD.

There is reasonable agreement between the overall ventilation flow rates and average zonal air temperatures predicted by the two modelling approaches. It is found that the multi-zone approach predicts a lower neutral level for the building than predicted by CFD. This might be mainly due to the fact that the thermal stratification is neglected in the present multi-zone model. It is explained by a new emptying air-filling box model for natural ventilation of single-zone buildings.

KEYWORDS

Natural ventilation, air flow pattern, modelling, temperature gradient, industrial buildings.

INTRODUCTION

Consider a single-zone enclosure with two openings at two different vertical levels (see Figure 1). A continuous heat source is located at the bottom level. A number of recent CFD simulations (Li *et al.* 1997) showed the basic physical process behind buoyancy-driven natural ventilation. The heat source first warms up the indoor air, which generates a stack pressure introducing incoming air through the lower opening and outgoing air through the higher opening. The incoming air is relatively cool. Being governed by negative buoyancy, the dense air tends to spread out at floor level. A plume flow is generated above the heat source. In the ambient outside the plume, a vertical temperature gradient and related density stratification is established. The vertical air temperature gradient affects not only the thermal plume development, but also the stack pressure which drives the ventilation air flow. Because the near-ceiling air is warmer than the near-floor air, the ceiling is also warmer than other surfaces. This gives rise to radiative heat transfer from the ceiling mainly to the floor. Thus, the natural ventilation process is governed by both fluid mechanics and heat transfer. This physical process is essentially the same as that in displacement ventilation (see for example Li *et al.* (1992)), except the inflow is governed by the thermal buoyancy force. This suggests that the vertical temperature profile in natural ventilation by thermal buoyancy can be calculated by the nodal models developed for displacement ventilation (for example Li *et al.* (1992)).

Most realistic buildings generally have multiple floors and multiple zones. Each zone can have a different air temperature from other zones and can be ventilated naturally in a dif-

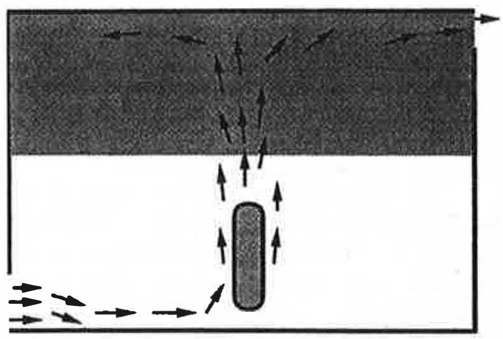

Figure 1 Basic principle behind buoyancy-driven natural ventilation in a single-zone building with two vertical openings and a single heat source.

ferent manner from other zones. In addition to some analytical methods, there are at least two major numerical modelling approaches for predicting air flow rate and air flow pattern, i.e. zonal methods and CFD methods.

The objective of this paper is to compare the prediction of ventilation flow rates through ventilation openings by both approaches and to investigate whether the thermal stratification should be considered in a zonal approach. The zonal approach assumes the flow in each zone is fully mixed. A realistic 10-zone industrial building is used as an example. A recently developed emptying air-filling box model for a single-zone building is used to explain the comparison results.

THE MODELS

Multi-zone model CHEETAH/MIX2.0

The MIX1.0 (Multi-cell Infiltration and eXfiltration) program was first developed by Li and Peterson (1990). In MIX1.0, an erroneous interpretation of the physical meaning of the concept of internal pressure in each zone prevented the calculation of bi-directional flows between zones. Li (1998) developed a consistent pressure-based formulation for natural ventilation of single-zone and multi-zone buildings with multiple openings. The formulation is made easier to implement by introducing an auxiliary concept of external pressure, which allows us to present all the formulas in a generalised form. The formulation was implemented in MIX2.0 (see Li *et al.* (1998)).

CHEETAH uses real hourly weather data to calculate temperatures in multi-zone buildings. It takes into account the thermal properties of materials (i.e. thermal resistance and capacitance), their areas and orientations, outdoor surface colours, glazing properties and so on. It can also take into account the air exchange between a zone and outdoors, and between zones.

The two programs were combined so that at each time-step (usually one hour or one minute), CHEETAH provides MIX2.0 with the zonal temperatures and calls MIX2.0 to obtain flow rates, and by iteration calculates a self-consistent set of flow rates and zonal temperatures. The combined program CHEETAH/MIX2.0 has been applied to a number of natural ventilation design problems.

CFD program Ventair

Our CFD program Ventair uses the conventional SIMPLE-family finite volume methods for discretising the three-dimensional flow equations with the Boussinesq approximation. The diffusion terms are discretised by the 2nd-order centred differencing scheme, while the convection terms are discretised by the 2nd-order QUICK scheme. In this paper, the standard $k - \varepsilon$ turbulence model is used.

Only the indoor domain is solved here. At an external opening, pressure is specified as:

$$p_0(z) = \frac{1}{2} \rho_0 C_p(z) v^2(z) \quad (1)$$

where ρ_0 is the outdoor air density. In all our calculations, the wind pressure coefficient $C_p(z)$ and wind velocity $v^2(z)$ are assumed to be uniform on an exterior surface.

The pressure at the first grid point near an opening, p_{int}, is calculated in the solution procedure.

When $p_0 > p_{int}$, there is an inflow, and the normal inflow velocity component is determined by:

$$v_{in} = \sqrt{\frac{2(p_0 - p_{int})}{\rho}} \quad (2)$$

When $p_0 < p_{int}$, there is an outflow, and the normal outflow velocity component is determined by:

$$v_{out} = \sqrt{\frac{2(p_{int} - p_0)}{\rho} + v_{int}^2} \quad (3)$$

The two tangential inflow or outflow velocity components are determined by the flow direction. A first reasonable approximation for the inflow direction is the wind direction and that for the outflow direction is the indoor air flow direction near the opening, i.e. a zero gradient boundary condition is used.

EMPTYING AIR-FILLING BOX MODEL

The question is how will neglecting thermal stratification affect the predicted air flow rates and neutral heights in a zonal approach. This can be studied by analytical solutions in a single-zone enclosure with two openings at two different vertical levels as discussed in the Introduction. Recently, two emptying air-filling box models were developed and other conventional models were revisited (Li 1998). Four models are considered here (see Figure 2). The major assumptions in these models can be summarised as follows:

- Fully mixed model – The model assumes that the indoor air temperature is uniform.
- Emptying water-filling box model – The enclosure is divided into two temperature zones, the bottom zone with cold and dense outside air, and the top with warm and lighter air. For simplicity, it is referred to here as the water model.
- Emptying air-filling box model I – The enclosure is also divided into two temperature zones; the bottom zone with cold and dense air, and the top zone with warm and lighter air. The bottom zone air can be heated by the warmer floor, which is heated up by surface radiation from the even warmer ceiling surface. The two air temperatures are calculated by the four-node model of Li *et al.* (1992). For simplicity, it is referred to as the air model.
- Emptying air-filling box model II – This model has the same assumption as model I, except that a linear vertical air temperature profile is assumed.

Ventilation flow rate, q, clean air zone height, h_c, and neutral height, z^*, can all be predicted by these four models, although some imagination is required to understand how a clean zone height can exist with the fully mixed model. The clean air zone height in this paper is simply defined as the height where the flow rate in the thermal plume equals the ventilation air flow rate, while the neutral height is defined as the height where the indoor and outdoor pressure difference is zero across the vertical wall. There are no simple analytical expressions for clean air zone height and neutral height for the emptying air-filling box model II.

The ventilation flow rate can be determined as follows.

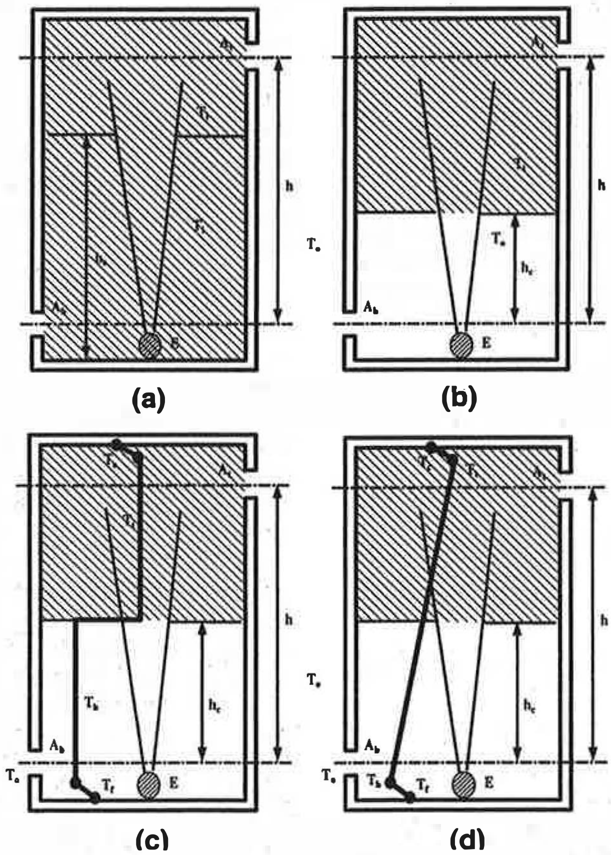

Figure 2 Basic assumptions in the four different emptying filling box models: (a) the fully mixed model; (b) the emptying water-filling box model; (c) the emptying air-filling box model I and (d) the emptying air-filling box model II.

- Fully mixed model:

$$q_1 = (C_d A^*)^{2/3} (Bh)^{1/3} \quad (4)$$

Emptying water-filling box model (the water model):

$$q_2 = (C_d A^*)^{2/3} [B(h - h_c)]^{1/3} \quad (5)$$

Emptying air-filling box model I (the air model):

$$q_3 = (C_d A^*)^{2/3} [B(h - h_c + \lambda h_c)]^{1} \quad (6)$$

Emptying air-filling box model II:

$$q_4 = (C_d A^*)^{2/3} \left[\frac{1}{2} B(1 + \lambda)\right]^{1/3} \quad (7)$$

where subscripts 1–4 indicate the type of the model; q is the ventilation flow rate; C_d is the discharge coefficient; B is the buoyancy flux, defined as $Eg/C_p \rho T_0$ with E being the heat output of the heat source; g is the gravitational acceleration; T_0 is the outdoor air temperature; h is the distance between the two ventilation openings; A^* is the effective opening area, defined as:

$$\frac{\sqrt{2}\, A_t A_b}{\sqrt{A_t^2 + A_b^2}}$$

and h_c is the clean zone height. λ is a non-dimensional number which is a function of ventilation flow rate, floor area, and convective and radiative heat transfer coefficients (see Li *et al.* (1992)).

Similarly, clean air zone heights can be calculated according to the following equations.

Let:

$$\xi = \frac{h_c}{h} \quad (8)$$

- Fully mixed model:

$$\frac{C_d A^*}{h^2} = C^{\frac{3}{2}} \sqrt{\xi_1^5} \quad (9)$$

- Emptying water-filling box model (the water model):

$$\frac{C_d A^*}{h^2} = C^{\frac{3}{2}} \sqrt{\frac{\xi_2^5}{1 - \xi_2}} \quad (10)$$

- Emptying air-filling box model I (the air model):

$$\frac{C_d A^*}{h^2} = C^{\frac{3}{2}} \sqrt{\frac{\xi_3^5}{1 - (1 - \lambda)\xi_3}} \quad (11)$$

where C is a constant (= 0.144).

The calculation of neutral heights is quite straightforward with these simple models.

Let:

$$\gamma = \frac{A_t^2}{A_t^2 + A_b^2} \quad (12)$$

- Fully mixed model:

$$z_1^* = \gamma h \quad (13)$$

- Emptying water-filling box model:

$$z_2^* = \gamma h + (1 - \gamma) h_c \quad (14)$$

- Emptying air-filling box model I:

$$z_3^* = \gamma h + (1 - \lambda)(1 - \gamma) h_c \quad (15)$$

A comparison of the first three models is shown in Figure 3 for an example building ($h = 6$ m and $T_0 = 293.15$ K) with equal ventilation openings ($\gamma = 0.5$) (Figures 3a–3c) and unequal openings areas ($\gamma = 0.2$) (Figure 3d). The buoyancy flux is $1.38e^{-2}$ m^4/s^4 (E = 500 W). C_d is 0.6. It seems that as the ventilation opening area approaches zero, all models give the same result. As the ventilation opening area increases, the difference between the results of the different models also becomes significant.

At $A/h^2 = 0.1$, the clean air zone height is the full building height for the fully mixed model due to very high ventilation flow rates (0.72 m^3/s). The water model predicts a result of 77% of the building height. With this model, the bottom zone has the same air temperature as outdoors. The indoor and outdoor air temperature difference in the bottom zone is zero and no ventilation air flow is introduced. This results in a much smaller ventilation flow rate through the ventilation openings (0.42 m^3/s). In displacement ventilation, the value of λ is generally around 0.4 (Mundt 1996). Results for three different λ values are shown in Figure 3 for the air model.

However, for neutral height predictions there is a more significant discrepancy between the results of the fully mixed model and the water model. While the fully mixed model predicts a constant neutral height for all ventilation openings, other models predict increasing heights as the ventilation opening areas increase. For equal ventilation openings, the water model predicts a neutral height of almost 90% of the full height,

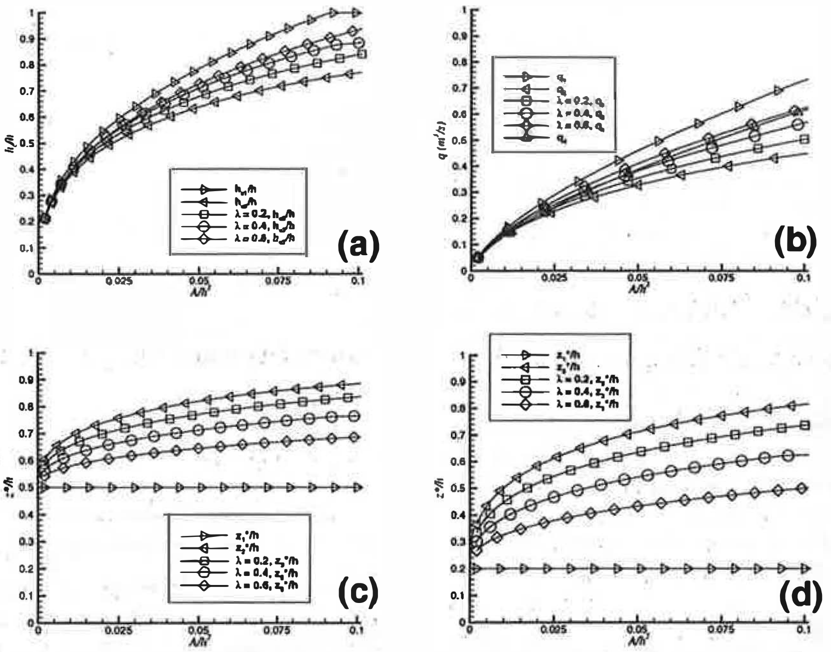

Figure 3 Comparison of predicted ventilation parameters by three emptying filling box models. The subscript 1 indicates the fully mixed model, 2 the emptying water-filling box model and 3 the emptying air-filling box model: (a) predicted clean air zone heights for $\gamma = 0.5$; (b) predicted ventilation flow rates for $\gamma = 0.5$; (c) predicted neutral levels for $\gamma = 0.5$ and (d) predicted neutral levels for $\gamma = 0.2$.

while the neutral height for the fully mixed model is only 50%. For the unequal area case, for the fully mixed model it is 20%, while the water model gives 80%. The air model results lie between those predicted by the fully mixed model and the water model.

THE BUILDING

The smelter building is naturally ventilated. Local exhaust ventilation systems are located above some large heat sources such as furnaces. The building is divided into ten zones (see Figure 4). Each zone represents an open space which is assumed to be well mixed. In the building considered here, the ventilation openings are typically doorways to outdoors, the open bottom around the building, floor gratings, open areas on each floor, drop zones through the floors, gaps around equipment such as furnaces, and the gap between the floors and the vertical walls. There are in total about 40 openings in the building. Only major ventilation openings are shown in Figure 4. The smelter also has a number of mechanical ventilation systems to draw off gaseous by-products etc.

The heat dissipation rates from the furnaces and equipment into the building were obtained from the building user. In most cases, it is fairly easy to distribute the heat to each zone. However, for two vertically connected zones, when the thermal plume above a heat source in the bottom zone penetrates through the opening between the two zones into the top zone, it is quite hard to distribute the heat in each zone in a multi-zone modelling program. In this paper, if the heat source is located fully in a zone, then its heat is assumed to be fully dissipated into that zone. There is a need to model the most important basic flow streams such as plumes and jets in a multi-zone approach, when the transport of heat and contaminants by plumes and jets is very significant. It should be mentioned that the conventional multi-zone approach only considers the flow governed by pressure differences, but not by momentum-introduced flows.

Our simulated results showed that the total heat gain in the building is largely dominated by the huge heat dissipation from the furnaces and equipment. Other sources including heat transmission from walls, heat storage in its structure and solar radiation contribute less than 10% of the total heat gain. Thus, for simplicity no description of detailed materials and wall thicknesses is given here.

RESULTS AND DISCUSSION

More than 40 multi-zone simulations were carried out to compare and evaluate the natural ventilation design, which provided a preliminary recommended design option. Then a number of CFD simulations were carried out to fine-tune the design. For the purpose of this paper, we will only compare the results for one case with no wind. Due to difficulties in resolving many exhaust hoods, CFD runs did not consider the exhaust flow of the mechanical system.

Flow pattern

The flow patterns predicted by CFD are shown in Figures 5 and 6. Strong thermal plumes are generated above furnaces and other hot equipment. These thermal plumes play a major role in carrying the heat upward and producing thermal stratification in the building. However, smaller thermal plumes generated above other heat sources in the furnace building interact strongly with inflow currents from wall ventilation openings. There is outflow through all the roof ventilators, although the outflow through the roof ventilator above the heat source is much stronger.

For the very large bottom openings around all of the building perimeter, the pressure boundary conditions used in this paper generate some very strange flows. The bottom openings are all 6.6 m high. Figure 7 shows the strange flow pattern generated at a height of 1.5 m. The heat sources behave as sinks and suck air in from all sides and carry it upward. Interaction of incoming flow streams results in stronger flows at corners, which seems to be unphysical (it was checked that the flow was fully converged and mass was conserved everywhere). For other locations, there can even be some outflows at the bottom (see also Figures 5 and 6). This may be physical, as the incoming air behaves like a gravity current in most situations, and air falls down to the floor (e.g. Figure 6) and spreads out along the floor. The falling air impinges on the floor, causing

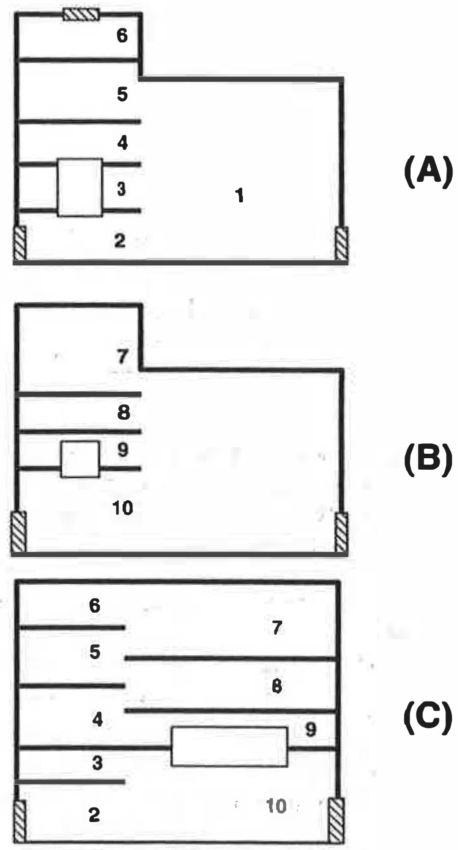

Figure 4 The geometry of the smelter building: (a) west end looking east; (b) east end looking west and (c) looking north.

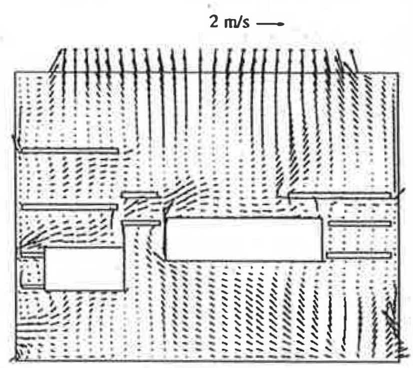

Figure 5 Velocity vector plot in the mid-plane cutting through the electric furnace. Only a quarter of the grid points are plotted for clarity.

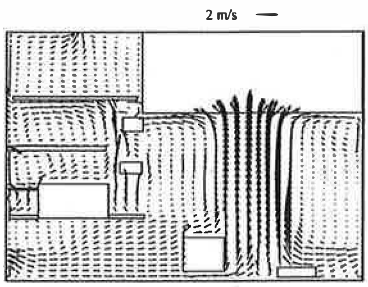

Figure 6 Velocity vector plot in the mid-plane cutting through the electric furnace. Only a quarter of the grid points are plotted for clarity.

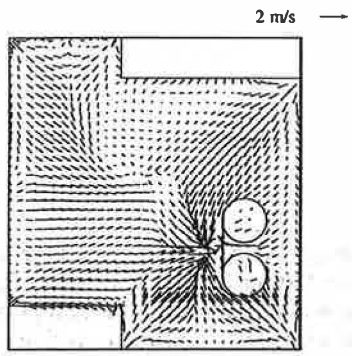

Figure 7 Velocity vector plot in a plane at 1.5 m above the floor of zones 1, 2 and 10. Only a quarter of the grid points are plotted for clarity.

air flow to be mainly forward, but some can be backward as well. The results suggest that more numerical studies are required to understand the types of boundary conditions used in this paper for natural ventilation with very large openings. Nevertheless, the present CFD predictions produced some reasonable flow patterns within the building.

Overall thermal and flow parameters

The averaged zonal temperatures and overall ventilation flow rates predicted by both methods are compared in Table 1.

When there is no wind, CFD predicted lower temperatures than the multi-zone method in the lower zones, while in upper zones CFD predicted higher temperatures. This is possibly due to thermal stratification in CFD simulations, which is not considered in the multi-zone computations. It should also be mentioned that in the CFD simulation performed here, surface radiation was not considered, which is also the case with the emptying water-filling box model discussed earlier.

As CFD runs did not include the exhaust flow of the mechanical ventilation systems, the CFD-predicted overall air flow rate (ACH) should be compared to the out air flow rate (exfiltration) in Table 1. There is fairly good agreement between the predicted overall flow rates by the multi-zone model (29 ACH) and CFD (26.5 ACH), although the CFD-predicted flow rate is lower than that predicted by the multi-zone model. The trend agrees well with the analytical results in Figure 3.

Table 1 Comparison of predicted air change rates and zonal air temperatures by a multi-zone model and CFD.

Zone	Zonal method	CFD
Zone 1 temp. (°C)	49.1	48.6
Zone 2 temp. (°C)	43.5	43.4
Zone 3 temp. (°C)	49.4	50.9
Zone 4 temp. (°C)	53.1	48.3
Zone 5 temp. (°C)	52.2	52.2
Zone 6 temp. (°C)	51.6	54.5
Zone 7 temp. (°C)	51.6	53.6
Zone 8 temp. (°C)	52.0	51.4
Zone 9 temp. (°C)	52.5	49.9
Zone 10 temp. (°C)	45.5	44.2
Infiltration (ACH)	34	
Exfiltration (ACH)	29	
Vent flow (ACH)		26.5

It is interesting that CFD predicted inflows through almost all wall openings for this building, while the multi-zone approach predicted outgoing flows through most upper wall openings. This situation is illustrated in Figure 8. CFD predicts a higher neutral plane than the multi-zone model. This can be attributed to the fact that the multi-zone model does not fully consider the thermal stratification in the building. It may be argued that multiple vertical zones partially account for the thermal stratification in part of the building. As shown in Figure 3 and in this section, although the overall ventilation flow rate can be predicted reasonably well by the zonal approach assuming fully mixed conditions, the neutral height is generally poorly predicted. This can be very significant for consideration of ventilation for local zones, for example in zones 4 and 5 in Figure 8.

CONCLUSIONS

For a ten-zone building naturally ventilated by thermal buoyancy, there is reasonable agreement between the predicted overall ventilation flow rates and average zonal air temperatures obtained by both the multi-zone model and the CFD model. However, the multi-zone approach predicts a much lower neutral level height for the building than that predicted by CFD analysis. This is due to the fact that thermal stratification is neglected in the present multi-zone model, and it is explained by a new emptying air-filling box model for natural ventilation of a single-zone building. Thermal stratification needs to be considered in a zonal approach for natural ventilation with large openings.

ACKNOWLEDGMENTS

The authors wish to thank H.H. Robertson Australia, who designed the natural ventilation system for the building, for the opportunity to perform the simulations described in this paper and for permission to publish some of the results. The authors also wish to thank Lester Rundle and Andrew Hripko from H.H. Robertson Australia for their useful discussions and supply of all necessary data for the simulation.

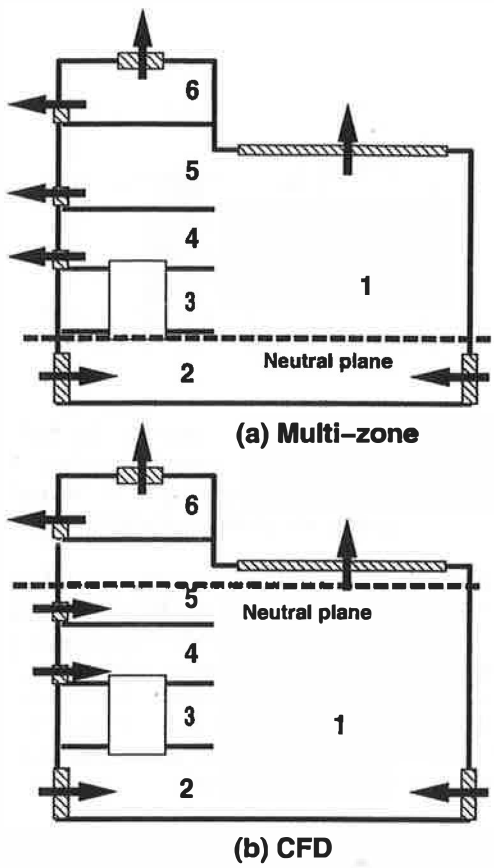

Figure 8 Illustration of predicted neutral levels by: (a) multi-zone model simulation and (b) CFD simulation.

REFERENCES

Delsante, A. (1987) CHEETAH – a thermal design tool for small buildings. CSIRO Division of Building Research Computer Manual, 87/1.

Li, Y. (1998) Prediction of natural ventilation in buildings with large openings: 1 – Basic formulations. Submitted to *Building and Environment*.

Li, Y., Delsante, A. and Symons, J.G. (1997) Simulation tools for analysing natural ventilation of large enclosures with large openings, *AIRAH Journal*, **52**(11), 22–28.

Li, Y., Delsante, A. and Symons, J. (1998) Prediction of natural ventilation in buildings with large openings: 2 – A computer implementation. Submitted to *Building and Environment*.

Li, Y. and Peterson, P. (1990) The modelling of multi-room airflow and its application to exhaust ventilation analysis. *Climate and Buildings*, KTH, no. 2, pp. 15–28.

Li, Y., Sandberg, M. and Fuchs, L. (1992) Vertical temperature profiles in rooms ventilated by displacement: full-scale measurement and nodal modeling. *Indoor Air*, **2**, 225–243.

Mundt, E. (1996) The performance of displacement ventilation systems – experimental and theoretical studies, PhD thesis, Royal Institute of Technology, Stockholm.