J. ENVIRONMENTAL SYSTEMS, Vol. 30(1) 45-72, 2003-2004

NUMERICAL STUDIES ON THERMALLY-INDUCED INDOOR AERODYNAMICS IN A COMPARTMENT WITH NATURAL VENTILATION

Q. KUI

W. K. CHOW

The Hong Kong Polytechnic University, China

S. L. LIU

Harbin Engineering University, China

ABSTRACT

Air flow induced by a point heat source in a natural ventilated compartment will be studied in this article. Three models, fully-mixed model, water-fully box model, and empty air-fully box model, on air flow in a chamber reported in the literature were reviewed. With a point heat source located at the center of the compartment, two different cases with and without walls were considered. A two-layer zone modeling approach with an upper hot air layer and a lower cool air layer is assumed. Two such zone models, Model 1 and Model 2, were developed: Model 1 is for a compartment without any walls. Model 2 is for an enclosed compartment with adiabatic walls, but with two openings: one at the top and the other at the bottom. Air flow induced by a point heat source in the natural ventilated compartments is solved. Volume flow rate equations and two-layer temperature difference equations were derived. Equations in the developed zone models were solved by the symbolic mathematics. The relationship between the neutral plane height and the outlet area was drawn. Results are compared with those predicted by Computational Fluid Dynamics. It is found that the two new models give results agreed fairly well with Computational Fluid Dynamics.

1. INTRODUCTION

Indoor air movement induced by thermal heat sources was studied to provide better ventilation design and more efficient smoke management systems [1]. There are numerous studies reported [2-9] in the literature. But only a few of these models are applicable to study air flow induced by a point heat source in a compartment. These included the water-filling box model by Linden [3], the fully-mixed model by Andersen [4], and emptying air-filling model by Li [9]. On the other hand, a two-layer model is commonly used for simulating fire [10]. In this article, indoor air flow induced by a point heat source was studied by a two-layer zone model. There is an upper hot air layer and a lower cool air layer with the plume serving as the vehicle to transfer heat and mass.

Two different cases with a point heat source located at the center of a compartment, with and without walls are considered:

- Model 1 is a compartment without any walls.
- Model 2 is an enclosed compartment with adiabatic walls, but two opening with one at the top and the other at the bottom.

With those models, air flow induced by a point heat source in a compartment can be studied. Equations in the developed zone models were solved by the symbolic mathematical packages [11, 12] with MATLAB [11] selected for the calculation. Results are compared with those by the Computational Fluid Dynamics (CFD) package PHOENICS [13].

2. A BRIEF REVIEW OF INDOOR AIR FLOW MODELS

There are many models developed for studying indoor air movement in a compartment with natural ventilation [1-9]. Three suitable models were selected with their key fluid dynamics equations [14, 15] briefly reviewed:

Fully-Mixed Model

In the fully-mixed model with a heat source in the chamber by Andersen [4], indoor air is assumed to be fully-mixed, the indoor air temperature is uniform, all the walls are adiabatic. Air inside is warmer than air outside.

A two-layer model is assumed due to the thermal source of power Q with the lower cool air at density ρ_1, temperature T_1; upper hot air at density ρ_2, temperature T_2; flow areas lower and upper parts are A_c and A_{c1} as in Figures 1 and 2. The volume flow rate V is given in terms of the flow coefficient C_d, and the separation height h as in Figure 1:

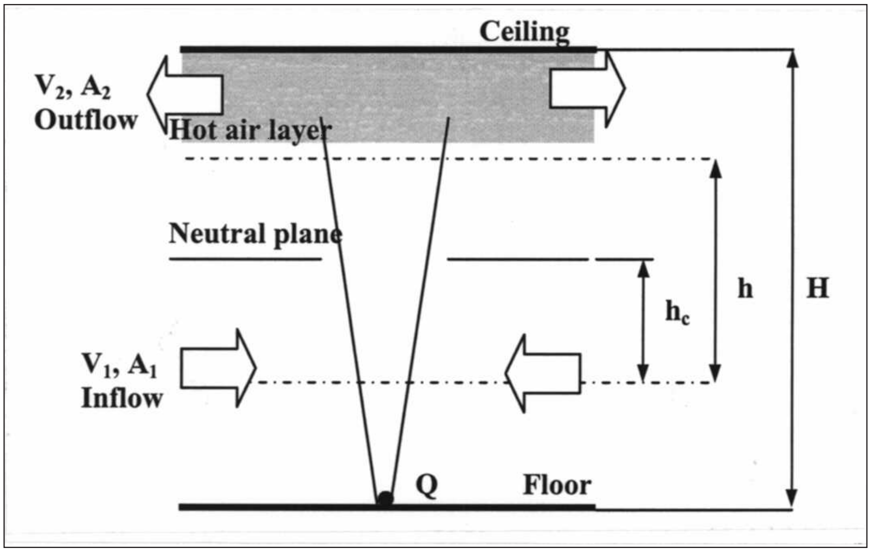


Figure 1. Geometry of Model 1.

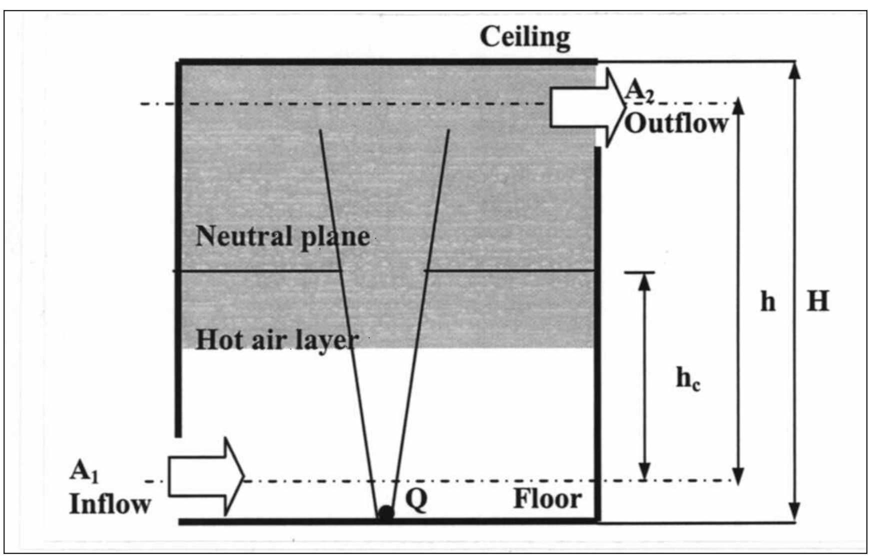


Figure 2. Geometry of Model 2.

$$V = C_d \sqrt{\frac{2A_1^2 A_2^2}{A_1^2 + A_2^2} \frac{\rho_2 - \rho_1}{\rho} gh} \quad (1)$$

Defining A as the effective opening area of the building:

$$A = \frac{\sqrt{2} A_1 A_2}{\left(\sqrt{A_1^2 + A_2^2}\right)} \quad (2)$$

The volume flow rate V is then:

$$V = C_d A \sqrt{\frac{T_1 - T_2}{T_2} gh} \quad (3)$$

Heat balance gives:

$$Q = \rho C_p V(T_1 - T_2) \quad (4)$$

Putting in equations (1) and (3):

$$T_1 - T_2 = \left(\frac{Q}{\rho_0 C_p C_d A}\right)^{2/3} \left(\frac{T_0}{gh}\right)^{1/3} \quad (5)$$

Defining the buoyancy flux B:

$$B = \frac{Qg}{C_p \rho_0 T_0} \quad (6)$$

V can be further simplified as:

$$V = (C_d A)^{2/3} (Bh)^{1/3} \quad (7)$$

Putting in the flow rate V_p calculated by the plume equation though a constant C, and the neutral plane height h_c:

$$V_p = C(Bh)^{1/3} h_c^{5/3} \quad (8)$$

Defining:

$$\xi = (h_c/h) \quad (9)$$

Equating the air entrainment rate to the plane V_p to ventilation flow rate V gives:

$$\frac{C_d A}{h^2} = C^{3/2} \sqrt{\xi^5} \quad (10)$$

Water-Filling Box Model

For the emptying water-filling box model by Linden [3], the compartment is also divided into two zones, the upper zone is lighter of density ρ_2 and higher temperature T_2 and the lower zone is with denser with density ρ_1 same as ambient and temperature T_1 fluid.

The volume flow rate V is also given by:

$$V = C_d A \sqrt{\frac{T_1 - T_2}{T_2} g(h - h_c)} \tag{11}$$

Heat balance gives:

$$T_1 - T_2 = \left(\frac{Q}{\rho_0 C_p C_d A} \right)^{2/3} \left(\frac{T_0}{g(h - h_c)} \right)^{1/3} \tag{12}$$

Putting equation (12) into (11):

$$V = (C_d A)^{2/3} (B(h - h_c))^{1/3} \tag{13}$$

After introducing the plume equation and equating the flow rate into plume with the ventilation rate,

$$\frac{C_d A}{h^2} = C^{3/2} \sqrt{\frac{\xi^5}{1 - \xi}} \tag{14}$$

Emptying Air-Filling Box Model

Surface thermal radiation effect in the building is included in the emptying air-filling box model by Li [9]. A constant temperature profile is assumed with the air temperature T_1 at the lower cool layer to be the same as the near-floor air temperature. Air temperature T_2 at the upper layer is the same as the air temperature next to the ceiling. This gives:

$$T_2 = \frac{Q}{\rho C_p V} + T_0 \tag{15}$$

$$T_1 = \lambda(T_2 - T_0) + T_0 \tag{16}$$

Note that λ is the temperature coefficient given by in terms of the convective heat transfer coefficient α_f at the floor; the radiative heat transfer coefficient α_r at the ceiling; and the convective heat transfer coefficient α_c at the ceiling.

$$\lambda = \left[\frac{\rho C_p V}{A_{floor}} \left(\frac{1}{\alpha_f} + \frac{1}{\alpha_r} + \frac{1}{\alpha_c} \right) + 1 \right]^{-1}$$

The volume flow rate V is:

$$V = C_d A \sqrt{\frac{T_1 - T_0}{T_0} g h_c + \frac{T_2 - T_0}{T_0} g (h - h_c)} \qquad (17)$$

Again, heat balance gives:

$$V = (C_d A)^{2/3} \, (Bh - h_c + \lambda h_c))^{1/3} \qquad (18)$$

and

$$\frac{C_d A}{h^2} = C^{3/2} \sqrt{\frac{\xi^5}{1 - (1 - \lambda)\xi}} \qquad (19)$$

3. THE TWO PROPOSED ZONE MODELS

Two zone models, Models 1 and 2, have been developed to study the phenomena of air movement in both an area of open boundary condition and an enclosed area.

Model 1

A compartment of height without walls in the perimeter as in Figure 1 is considered. The dimension of the room is of length L, width W, and height H. A point heat source is located at the center of the chamber. A thermally-driven plume would rise to the ceiling, forming a ceiling jet and then a hot air layer afterward. The lower part is considered the cool air layer. Eventually, ambient air would be drawn into the chamber, moving into the cool air layer, being heated up and rise to the hot air layer, and then flow out of the chamber through the hot layer.

Model 2

This is a compartment of the same size as in Model 1 but surrounded by adiabatic walls. There are two openings, one at the top and one at the bottom. A point heat source is located at the center of the chamber as in Figure 2. Again, a two-layer zone model is assumed. Ambient air would flow into the chamber through the lower opening, heated up, rise to the ceiling, and then flow out through the upper opening.

The equations for volume flow rates inflow V_1 and outflow V_2 are derived in Appendix A and summarized as follows:

$$V_1 = C_{d1} A_1 \sqrt{\frac{2\Delta T g h_c}{T_2}} \tag{20}$$

$$V_2 = C_{d2} A_2 \sqrt{\frac{2\Delta T g (h - h_c)}{T_0}} \tag{21}$$

The buoyancy flux B is given by:

$$B = \frac{Qg}{C_p \rho_0 T_0}$$

Equations (20) and (21) become:

$$V_1 = (C_{d1}A_1)^{2/3}\ (2Bh_c)^{1/3} \tag{22}$$

$$V_2 = (C_{d2}A_2)^{2/3}\ (2B(h - h_c))^{1/3} \tag{23}$$

The temperature difference between the upper layer and the lower layer is:

$$\Delta T = \left(\frac{Q}{\rho_0 C_p C_{d2} A_2}\right)^{2/3} \left(\frac{T_0}{2g(h - h_c)}\right)^{1/3} \tag{24}$$

The neutral level h_c is defined as the height where the pressure difference across the opening is zero and is given by:

$$h_c = \frac{h}{n + 1} \tag{25}$$

where

$$n = \frac{T_2}{T_0}\left(\frac{C_{d1} A_1}{C_{d2} A_2}\right)^2$$

The relationship between the neutral level height h_c and the outlet area A_2 is:

$$\frac{A_2}{h^2} = 0.042318 / C_{d2} \sqrt{\frac{\left(\frac{h_c}{h}\right)^5}{\left(1 - \frac{h_c}{h}\right)}} \tag{26}$$

4. ZONE MODELING RESULTS

Results predicted by the two developed zone model are solved by MATLAB [11] with listing shown in Appendix B. Data in some examples were analyzed by Maple-V [12] as listed in Appendix C.

Ventilation flow rates predicted by Model 1 and Model 2 are compared with the emptying air-filling box model by Li [9] given by equation (18). Results are as shown in Figure 3 and compared with equation (23) in Model 2. The relationship between the outlet area A_2, the distance between the inlet and outlet opening h and the outlet volume ventilation flow rate V_2 is shown the figure. It can be seen that the outlet flow rate would increase when A_2/h^2 is increased. There is a very small difference between the results of the two models for A_2/h^2 less than 0.02.

The neutral plane heights of Model 1 and Model 2 are plotted in Figure 4 for comparison with the emptying air-filling box model ($\gamma = 0.2$). There is an equation on h_c in the emptying air-filling box model given by:

$$h_c = \gamma h + (1 - \lambda)(1 - \gamma)h_1 \qquad (27)$$

The above equation is compared with equation (26) in Model 2. Note that γ is the vertical temperature gradient. The relationship between the neutral zone

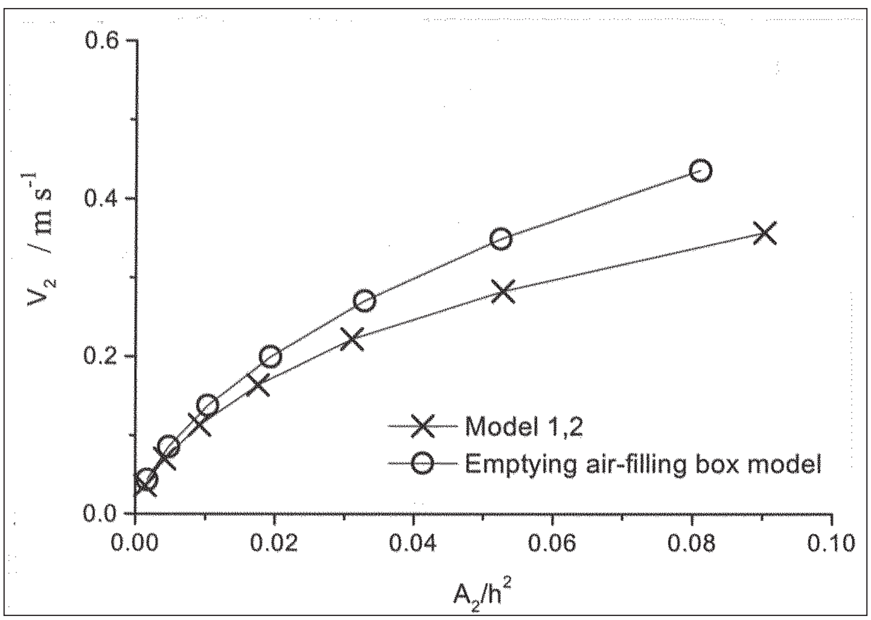


Figure 3. Outlet ventilation flow rates.

height h_c, the distance between the inlet and outlet opening h, and the outlet area A_2 is shown in Figure 4. The result showed that the height of the neutral plane would increase when A_2/h^2 is increased. For A_2/h^2 less than 0.02, the neutral plane height increased. There are large differences between the results of the two models. But for A_2/h^2 greater than 0.02, neutral plane heights predicted by the two models agreed better.

For interest, the neutral plane heights predicted by Model 1 and 2 are compared with the hot air layer height of the emptying air-filling box model in Figure 5 to show the similarities of the two parameters in the two models. The following equation in the emptying air-filling box model is compared with equation (26):

$$\frac{C_d A_2}{h^2} = C^{3/2} \sqrt{\frac{\left(\frac{h_1}{h}\right)^5}{\left(1-(1-\lambda)\frac{h_1}{h}\right)}} \tag{28}$$

where C is a constant used in the plume flow rate calculation.

The result showed that the neutral plane height h_c in Model 1 and Model 2 agrees well with the hot air layer height in the emptying air-filling box model when the temperature coefficient λ is 0.2.

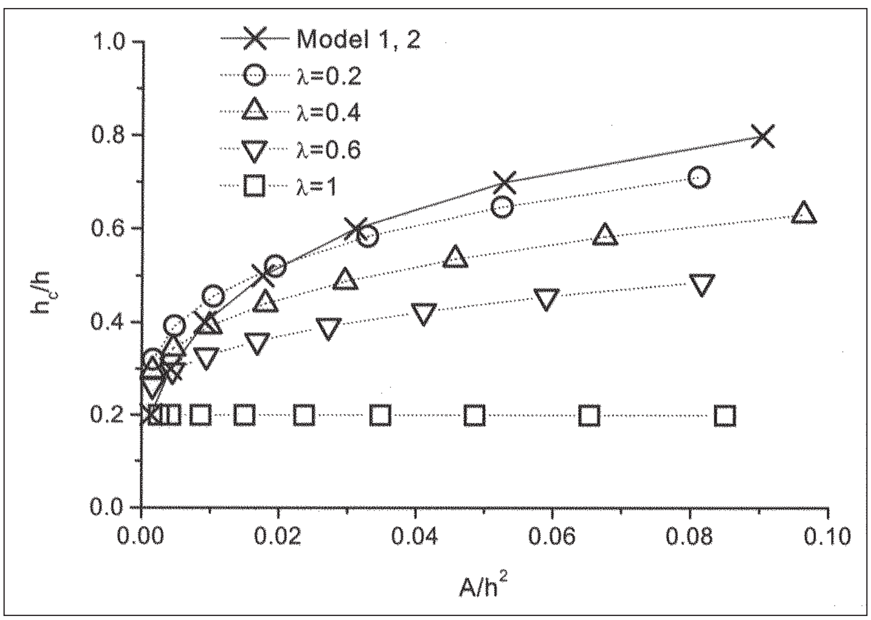


Figure 4. Neutral plane height for $\gamma = 0.2$.

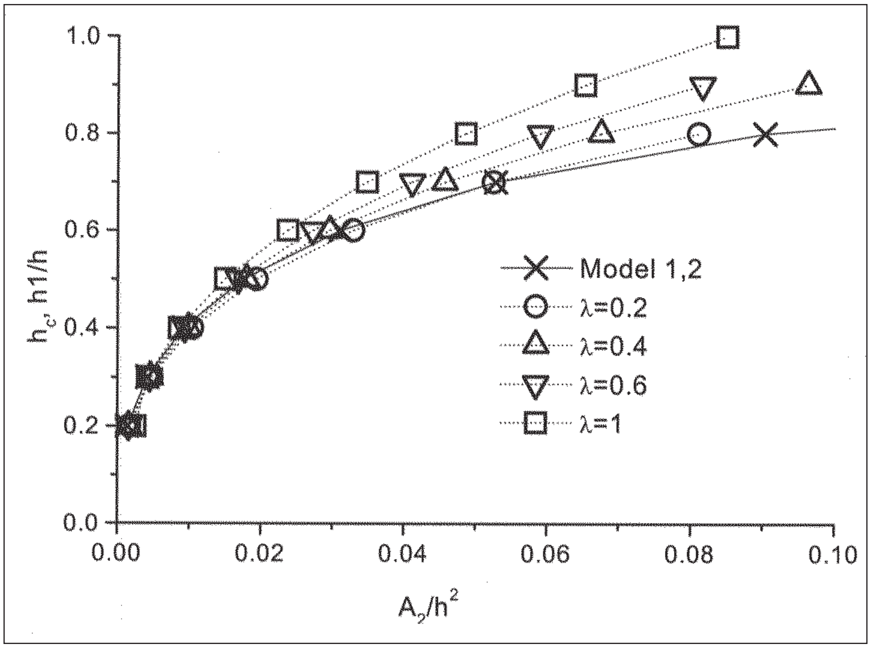


Figure 5. Neutral plane height (h_c/h) of Model 1 and Model 2 with the hot air layer height (h_1/h) of emptying air-filling box model.

Comparisons are also made in the following conditions to study the inflow and outflow rates in the zone model separately. The height of the chamber H is taken as 6 m with both the upper and lower openings in Model 2 to be 1 m^2. A point heat source is located at the center of the control volume, with a constant heat release rate of 500 W. The ambient temperature T_0 is 300 K. To simplify the calculation, C_d is 0.6, C_p is 1010 $Jkg^{-1}K^{-1}$, and p_0 is 101300 Pa.

A comparison of the inlet airflow rate in Model 1 and Model 2 with Andersen's model is shown in Figure 6. The following equation in Anderson's model [4] is compared with equation (22):

$$V_1 = 0.037\,(Qh_c)^{1/3}\,(C_{d1}A_1)^{1/2} \tag{29}$$

Results showed that the inflow rate V_1 increased when the neutral plane height h_c increased.

A comparison of the outlet airflow rate in Model 1 and Model 2 with Andersen's model is shown in Figure 7. The following equation in Andersen's model [4] is compared with equation (23):

$$V_2 = 0.038(Q(h - h_c))^{1/3}\,(C_{d2}A_2)^{1/2} \tag{30}$$

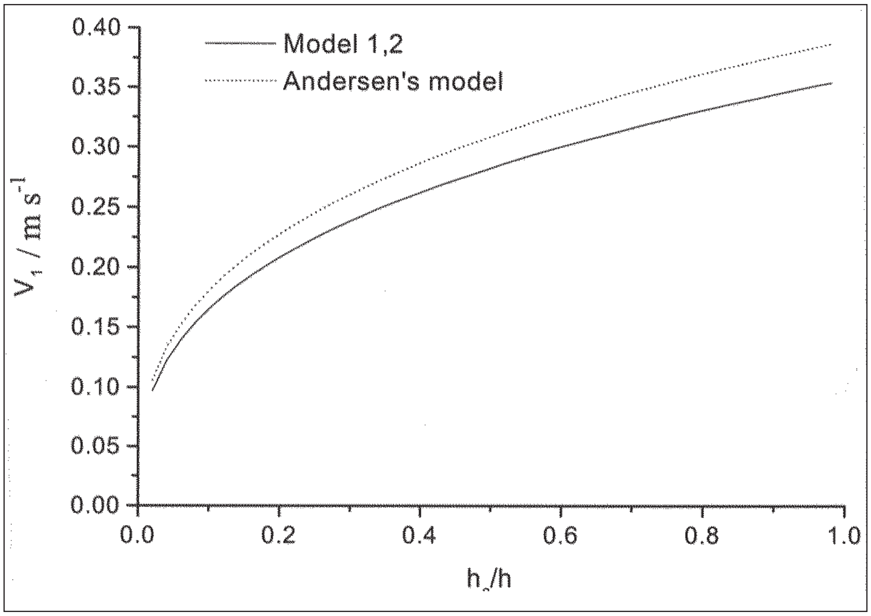


Figure 6. Inlet airflow rates.

The results showed that when the neutral plane height h_c increased, the outflow rate V_2 decreased.

Finally, a comparison of the temperature difference in Model 1 and Model 2 with Andersen's model is made in Figure 8. The equation in Andersen's model [4] is compared with equation (24):

$$\Delta T = 7.5 \cdot 10^{-5} T_0 \left(\frac{Q}{C_{d2} A_2} \right)^{2/3} \left(\frac{1}{h - h_c} \right)^{1/3} \tag{31}$$

It is found that before the neutral plane height $h_c/h = 0.8$, there are less changes in the temperature difference ΔT of the two layers, but after that point there is an obvious increase in the temperature difference of the two layers with the increase of h_c/h.

The relationship between the neutral zone height h_c, height of inlet and outlet opening h and the outlet area A_2 is shown by equation (26). But usually, the area and position of the openings are known first, and the neutral zone height needs to be solved. That could be a very complex equation but can be fitted by:

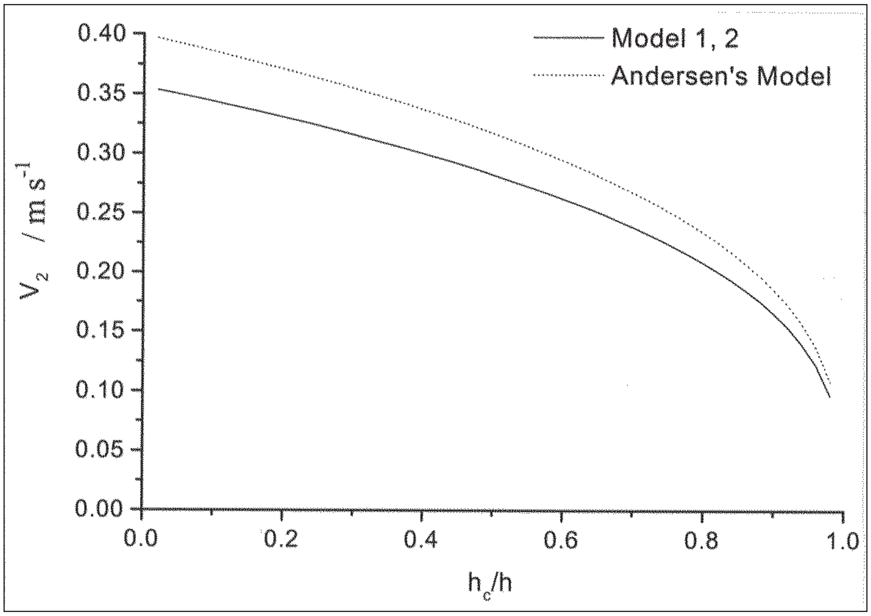


Figure 7. Outlet airflow rate.

$$\frac{h_c}{h} = 1.7276\left(\frac{A_2}{h^2}\right)^{0.3187} \tag{32}$$

Results are plotted in Figure 9 with the corresponding coefficient being 0.9899.

5. SIMULATION WITH COMPUTATIONAL FLUID DYNAMICS

The Computational Fluid Dynamics (CFD) was used for comparing the zone modeling results. The commercial software PHOENICS [13] was taken as the simulator. A compartment with L of 10 m, W of 10 m, and H of 6 m was taken for comparison of the two models. The grid and geometry of CFD simulation are shown in Figure 10.

Airflow vectors predicted are shown in Figure 11 to show the air movement pattern. Maximum air speed changed from 0.187 ms^{-1} in the open boundary condition to 0.2 ms^{-1} in the enclosed condition. The inflow rate and outflow rate were calculated from the results of CFD simulation.

The simulation results of pressure distribution are shown in Figure 12. The height of neutral plane is clearly shown. The result of CFD simulation and the result of the zone models are compared in Table 1.

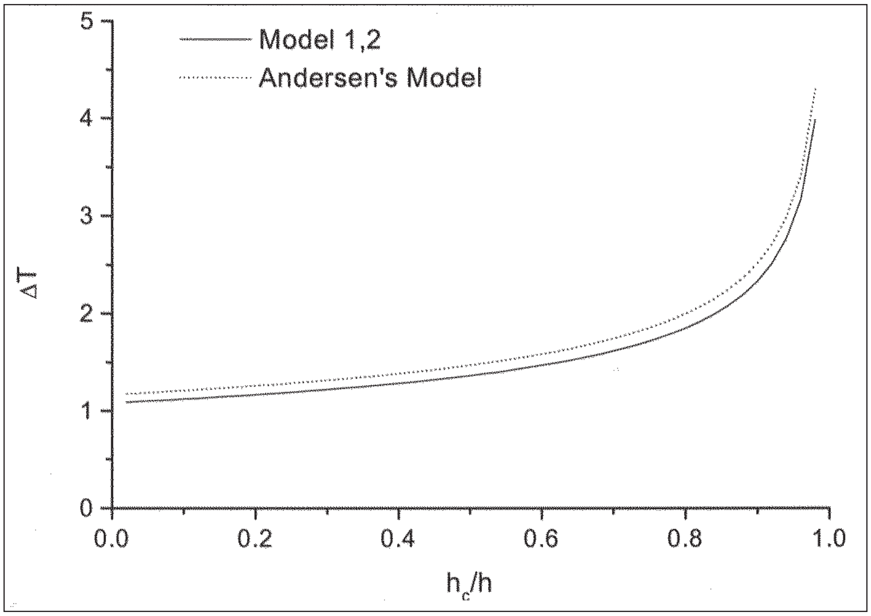


Figure 8. Temperature difference.

The results showed that the inflow rate and outflow rate in CFD simulation are very close, but the results in the theoretical zone model simulation are quite different. In Model 1, the outflow ventilation rate is limited by the small cross-sectional area of outflow air. In Model 2, the outlet area and the inlet area are the same, but the neutral plane has changed a little, and the outflow rate is still limited by the theoretical zone model equation. The inflow and outflow results in Model 2 in CFD simulation are quite different from the zone model result. That is because only the air flowing in the control volume through A_1 and flowing out through A_2 was considered. It is assumed that each zone is fully mixed. The turbulence inside the chamber was not considered. But from Figure 10, it is obvious to see that most of the air would rise to the ceiling and create turbulence inside the chamber, so that the inflow rate and outflow rate are significantly reduced to a lower level.

6. CONCLUSION

Two zone models are developed to study air flow problems induced by a point heat source in a natural ventilated chamber. The difference in the results of CFD simulation [13] and zone model simulation was discussed. Model 1 could be considered as a special situation of Model 2, when the area of the upper opening

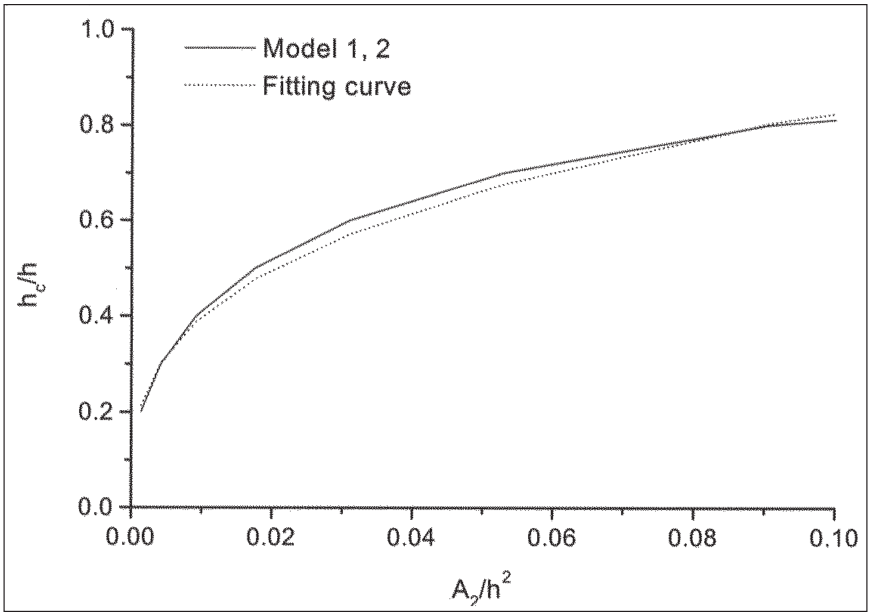


Figure 9. Fitting of the neutral plane equation.

and the lower opening in Model 2 is exactly equal to the opening area of upper layer and lower layer in Model 1 respectively.

Volume flow rate equations and two-layer temperature difference equations were derived. The relationship between the neutral level height and the outlet area was drawn. The neutral plane h_c equation was defined by curve-fitting, which made the solution easy to use.

In comparing with CFD results, the new models give fairly good predictions, as seen in Table 1.

Table 1. A Comparison between the Results of CFD Simulation and the Two Zone Models

	CFD simulation		Zone model	
Parameters	Model 1	Model 2	Model 1	Model 2
Inflow rate V_1/m^3s^{-1}	1.57	7.16×10^{-2}	0.35	0.30
Outflow rate V_2/m^3s^{-1}	1.47	8.00×10^{-2}	0.13	0.25
Neutral plane h_c/m	3.58	3.58	2.9	3.1
Neutral plane height from floor/m	3.58	3.58	5.8	3.6

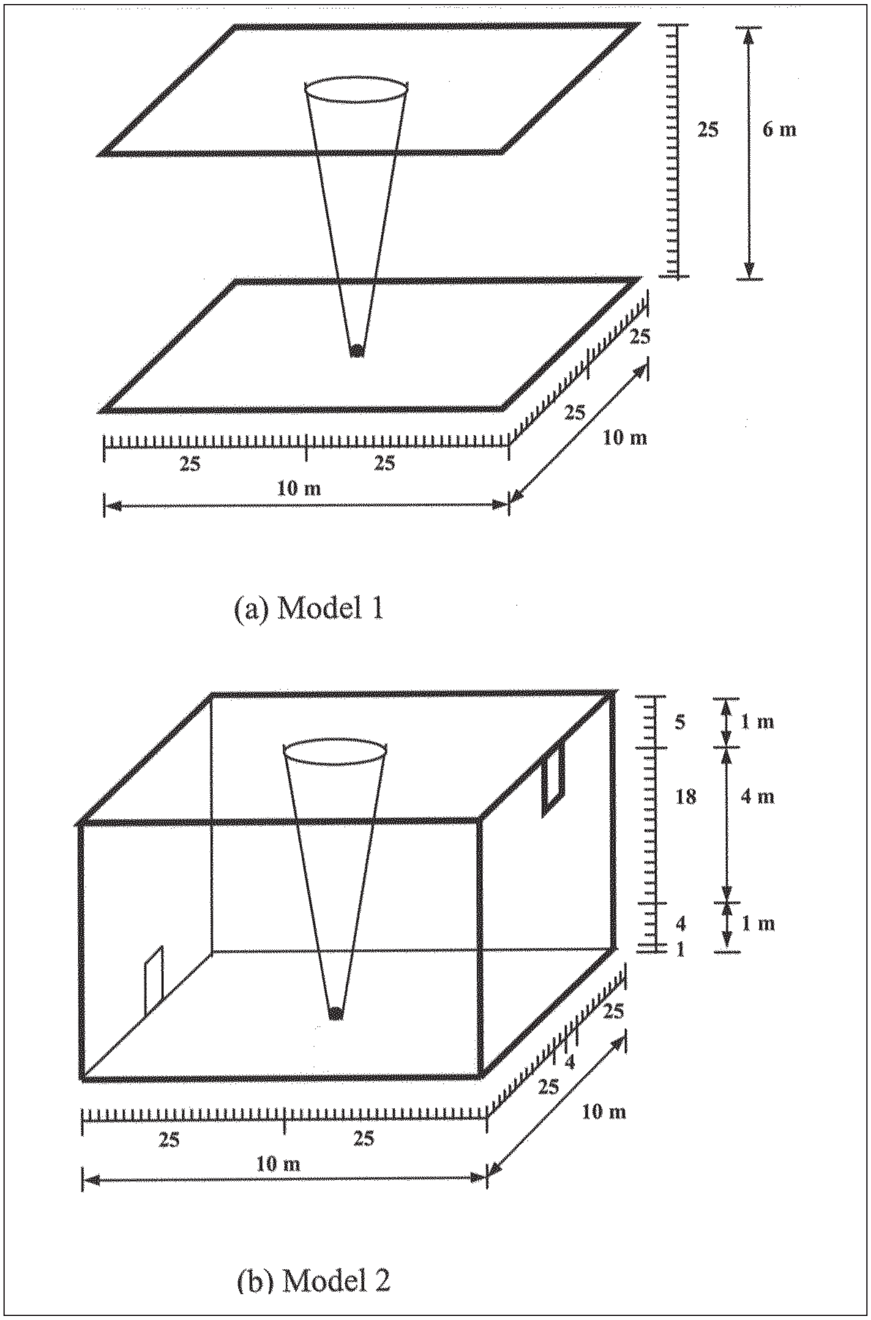
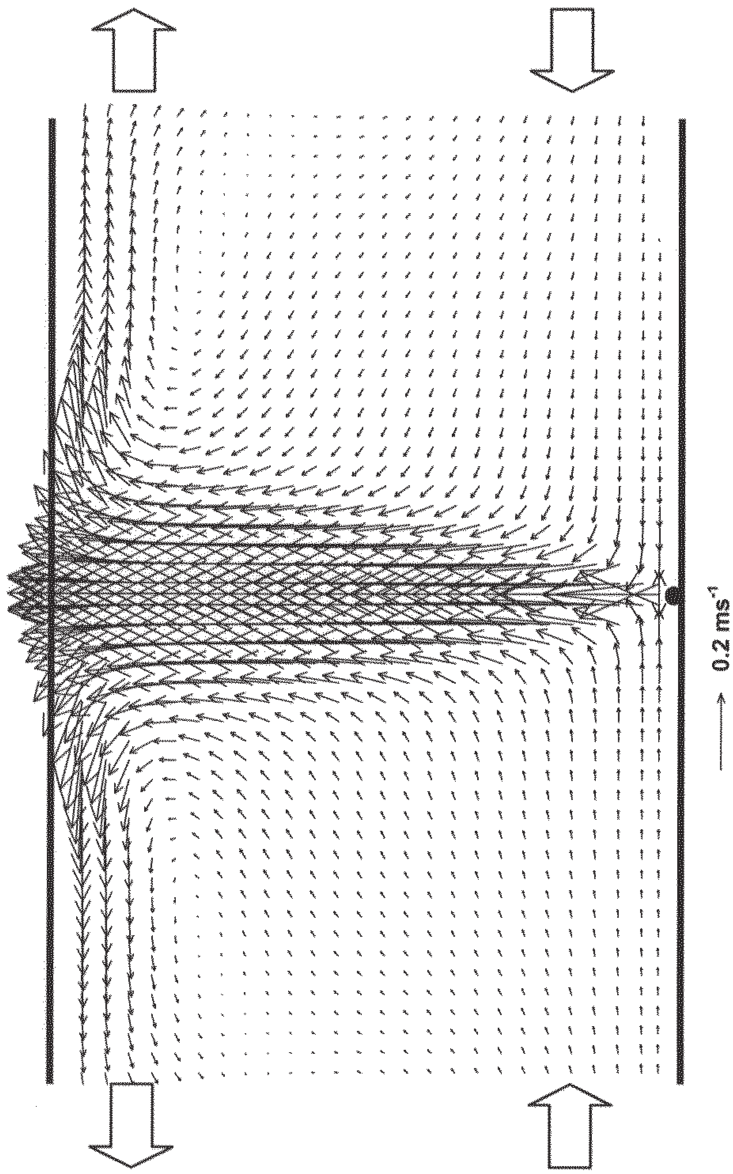
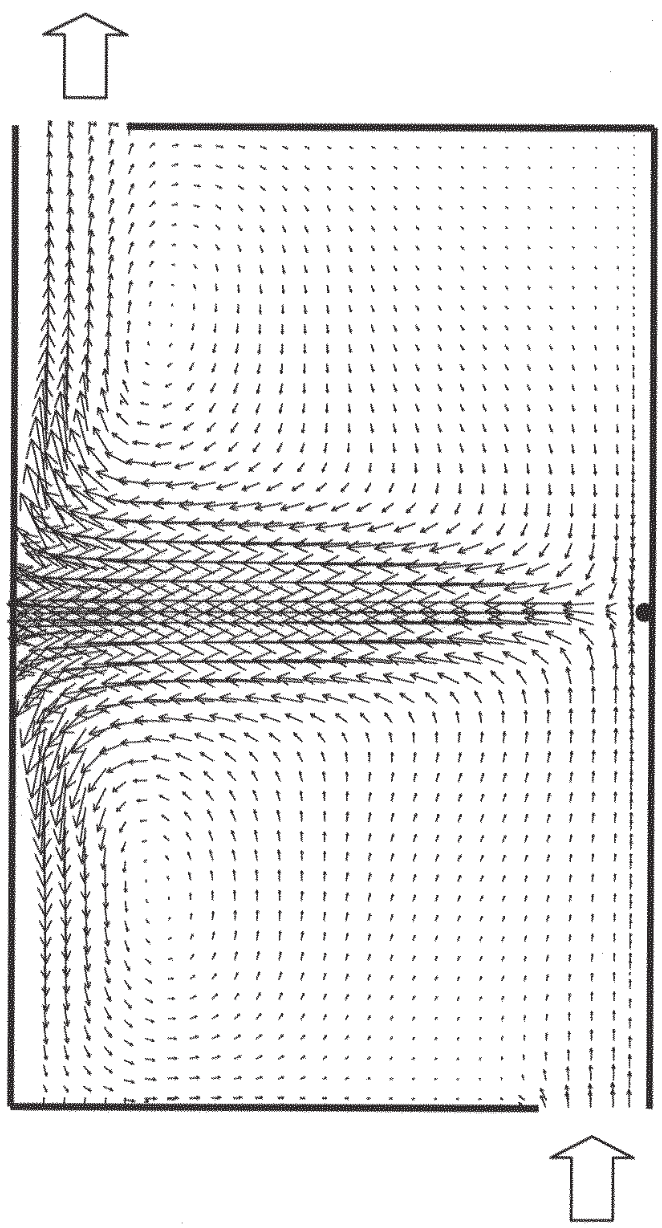


Figure 10. Geometry and grid of the CFD simulation.

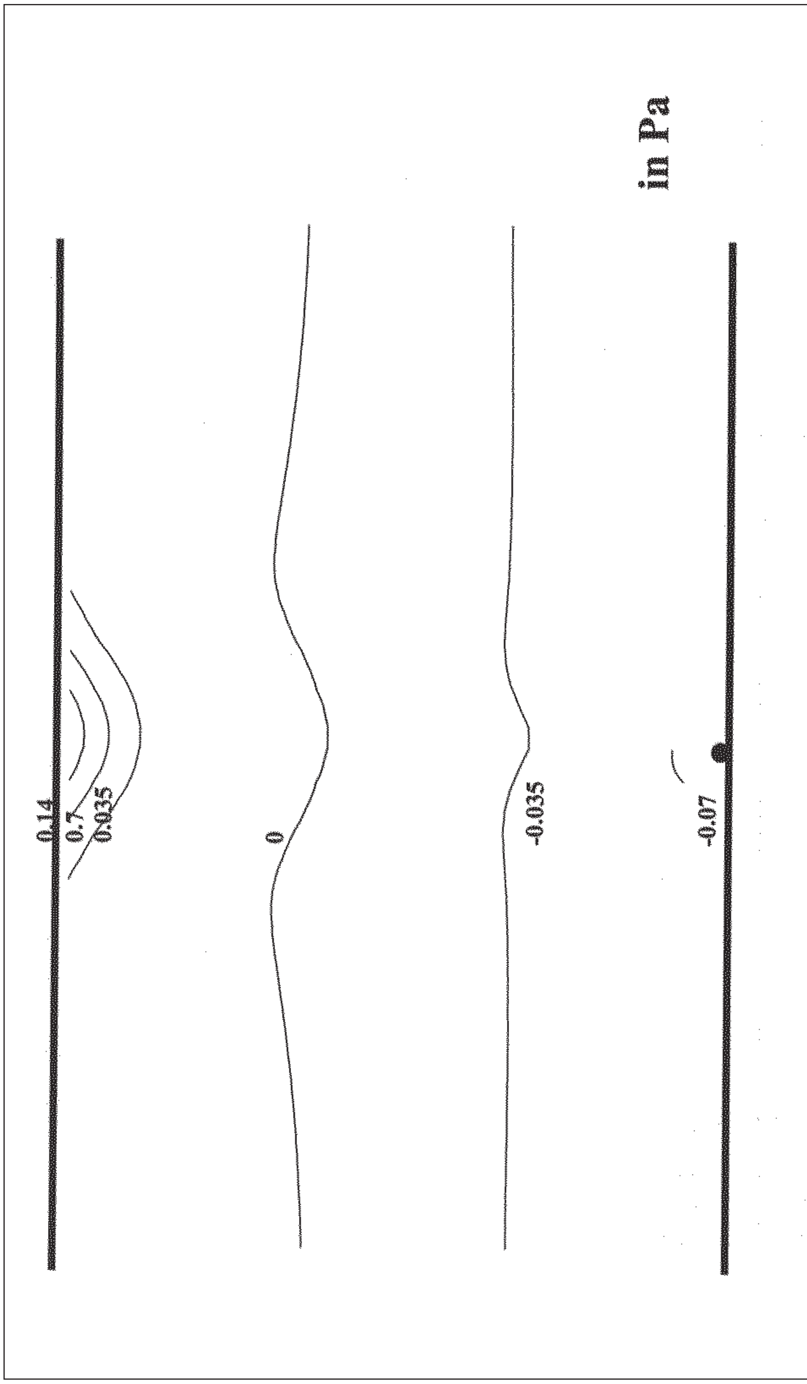


(a) Model 1

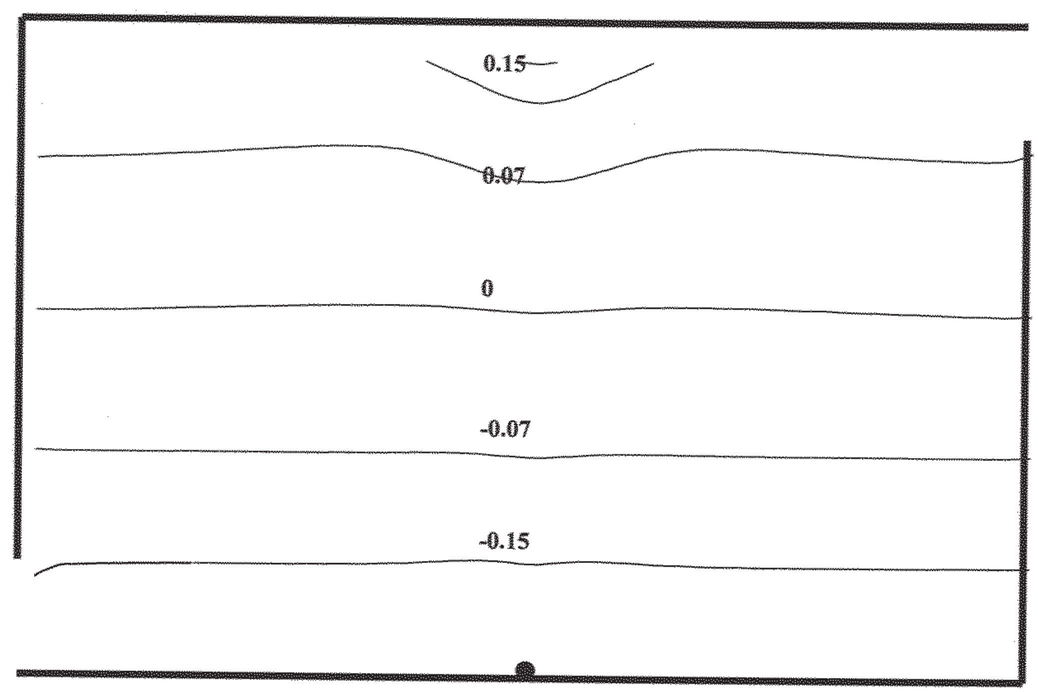


(b) Model 2

Figure 11. Vector results in CFD simulation.



(a) Model 1



(b) Model 2

Figure 12. Pressure distribution results in CFD simulation.

APPENDIX A: Derivation of Key Equations in the Zone Model

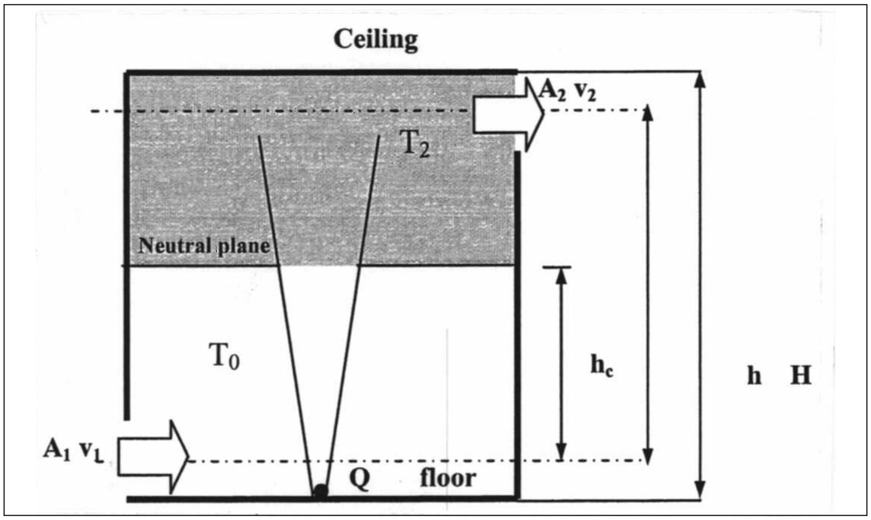


Figure A1. Two-layer zone model.

The two-layer zone model is shown in Figure A1. The point heat source is taken as at the center on the floor. The following assumptions were made:

- The rate of entraimnent at the edge of the plume is proportional to some characteristic velocity at that height.
- The profiles of mean vertical velocity and mean buoyancy force in horizontal sections are of similar form at all heights.
- The largest local variations of density in the field of motion are small in comparison with some chosen reference of density, this reference being taken as the density of the ambient fluid at the level of the source.
- The scale of the initial laminar zone is small compared with the subsequent zone of turbulent convection.
- The pressure is hydrostatically distributed throughout the field of motion.
- Transverse forces are ignored in comparison with those in the vertical direction.
- Turbulent mixing in the vertical direction is ignored in comparison with that in the horizontal.
- No mixing between the plume and ambient air flow.
- Floor, ceiling, and walls of the chamber are made of adiabatic materials.

Under the assumptions above, the mass balance equation of zone model is:

$$\rho_0 A_{c1} v_{c1} = \rho_2 A_{c2} v_{c2} \quad \text{(A1)}$$
$$A_{c1} = C_{c1} A_1$$
$$A_{c2} = C_{c2} A_2$$

Energy equation is:

$$Q = \rho_0 C_p A_{c2} v_{c2} \Delta T \quad \text{(A2)}$$
$$\Delta T = T_2 - T_0$$

h_c is defined as the height of the neutral plane.

$$\Delta p_1 = \Delta \rho g h_c$$

$$\Delta p_2 = \Delta \rho g (h - h_c)$$

$$\frac{\Delta p_1}{\rho_0} = \frac{1}{2} \xi v_{c1}^2$$

$$v_{c1} = \sqrt{\frac{2\Delta p_1}{\rho_0}} = \sqrt{\frac{2\Delta \rho g h_c}{\rho_0 \zeta}} \quad \text{(A3)}$$

$$\frac{\Delta p_2}{\rho_2} = \frac{1}{2} \zeta v_{c2}^2$$

$$v_{c2} = \sqrt{\frac{2\Delta p_2}{\rho_0}} = \sqrt{\frac{2\Delta \rho g (h - h_c)}{\rho_0 \zeta}} \quad \text{(A4)}$$

The volume flow rate equation for inflow is:

$$V_1 = A_{c1} v_{c1} = C_{c1} A_1 \sqrt{\frac{2\Delta \rho g h_c}{\rho_0 \zeta}} = C_{d1} A_1 \sqrt{\frac{2\Delta \rho g h_c}{\rho_0}} \quad \text{(A5)}$$

where $C_{d1} = C_{c1}/\xi^{1/2}$

and the volume flow rate equation for outflow is:

$$V_2 = A_{c2} v_{c2} = C_{c2} A_2 \sqrt{\frac{2\Delta \rho g (h - h_c)}{\rho_2 \zeta}} = C_{d2} A_2 \sqrt{\frac{2\Delta \rho g (h - h_c)}{\rho_2}} \quad \text{(A6)}$$

where $C_{d1} = C_{c1}/\xi^{1/2}$

From equations (A1) to (A4), it is simply known that

$$\frac{\rho_0}{\rho_2} = \frac{T_2}{T_0} = \frac{V_2}{V_1} \qquad \frac{\Delta T}{T_0} = \frac{\Delta \rho}{T_2} \qquad \frac{\Delta T}{T_2} = \frac{\Delta \rho}{\rho_0}$$

$$\Delta T = T_2 - T_0$$

$$V_1 = C_{d1} A_1 \sqrt{\frac{2\Delta T g h_c}{T_2}} \tag{A7}$$

$$V_2 = C_{d2} A_2 \sqrt{\frac{2\Delta T g (h - h_c)}{T_0}} \tag{A8}$$

From equation (A2),

$$\Delta T = \frac{Q}{\rho_0 C_p V_2} = \frac{Q}{\rho_0 C_p C_{d2} A_2 \sqrt{\frac{2\Delta T g (h - h_c)}{T_0}}}$$

$$\Delta T^3 = \frac{Q^2 T_0}{(\rho_0 C_p C_{d2} A_2)^2 \, 2g(h - h_c)}$$

The temperature difference between the upper layer and the lower layer is:

$$\Delta T = \left(\frac{Q}{\rho_0 C_p C_{d2} A_2} \right)^{2/3} \left(\frac{T_0}{2g(h - h_c)} \right)^{1/3} \tag{A9}$$

Defining the buoyancy flux B:

$$B = \frac{Qg}{C_p \rho_0 T_0}$$

$$T_2 = \frac{Q}{\rho_0 C_p V_2} + T_0$$

$$V_2 = C_{d2} A_2 \sqrt{\frac{2Qg(h - h_c)}{\rho_0 C_p V_2 T_0}}$$

$$V_2{}^3 = \left(C_{d2} A_2 \right)^2 \frac{2Qg(h - h_c)}{\rho_0 C_p T_0}$$

$$V_2 = (C_{d2} A_2)^{2/3} \, (2B(h - h_c))^{1/3} \tag{A10}$$

Similarly,

$$V_1 = (C_{d1} A_1)^{2/3} \, (2Bh_c)^{1/3} \tag{A11}$$

The neutral level h_c is defined as the height where the pressure difference across the building envelope is zero.

From equation (A1),

$$\rho_0 V_1 = \rho_2 V_2$$

Combining with equations (A7) and (A8),

$$\rho_0 C_{d1} A_1 \sqrt{\frac{2\Delta T g h_c}{\mathrm{T}_2}} = \rho_2 C_{d2} A_2 \sqrt{\frac{2\Delta T g (h - h_c)}{T_0}}$$

$$(\rho_0 C_{d1} A_1)^2 T_0 \, 2\Delta T g h_c = (\rho_2 C_{d2} A_2)^2 T_2 \, 2\Delta T g (h - h_c)$$

$$h_c = \frac{(\rho_2 C_{d2} A_2)^2 T_2 h}{(\rho_0 C_{d1} A_1)^2 T_0 + (\rho_2 C_{d2} A_2)^2 T_2}$$

$$h_c = \frac{h}{\frac{T_2}{T_0}\left(\frac{C_{d1} A_1}{C_{d2} A_2}\right)^2 + 1} = \frac{h}{n+1} \qquad \text{(A12)}$$

where

$$n = \frac{T_2}{T_0}\left(\frac{C_{d1} A_1}{C_{d2} A_2}\right)^2$$

$$V_1 = (C_{d1} A_1)^{2/3} (2Bh_c)^{1/3}$$

$$V_2 = (C_{d2} A_2)^{2/3} (2B(h - h_c))^{1/3}$$

The equation of thermal plume above a fire was developed by Rouse and Yih [14, 15]:

$$M = 0.153\rho_2 \left[\frac{Qg}{(\rho_2 C_p T_0)}\right]^{1/3} (h - (h - h_c))^{5/3}$$

$$M = 0.153\rho_2 \left[\frac{Qg}{(\rho_2 C_p T_0)}\right]^{1/3} h_c^{5/3} \qquad \text{(A13)}$$

$$M = \rho_2 A_2 v_2 = \rho_2 V_2$$

$$V_2 = \frac{M}{\rho_2} = \frac{0.153\rho_2 [Qg / (\rho_2 C_p T_0)]^{1/3} h_c^{5/3}}{\rho_2} = 0.153 B^{1/3} h_c^{5/3} \qquad \text{(A14)}$$

Combining equations (A10) and (A14),

$$V_2 = (C_{d2}A_2)^{2/3}\,(2Bh(h-h_c))^{1/3} = 0.153B^{1/3}h_c^{5/3}$$

$$(C_{d2}A_2)^2\,2(h-h_c) = 0.153^3 h_c^5$$

$$C_{d2}A_2 = \sqrt{\frac{0.153^3\,h_c^5}{2(h-h_c)}}$$

$$\frac{C_{d2}A_2}{h^2} = \sqrt{\frac{0.153^3\left(\frac{h_c}{h}\right)^5}{2\left(1-\frac{h_c}{h}\right)}}$$

$$\frac{C_{d2}A_2}{h^2} = 0.042318\sqrt{\frac{\left(\frac{h_c}{h}\right)^5}{\left(1-\frac{hc}{h}\right)}} \qquad \text{(A15)}$$

That is the relationship between the neutral level height h_c and the outlet area A_2.

APPENDIX B: MATLAB Listing

```
dt=zeros (1, 9);
V=zeros (1, 9) ;
y1=zeros (1, 9);
y2=zeros (1, 9);
y3=zeros (1, 9);
h1=0;
i=0;
j=0;
A1=1;
A2=1;
e=0.1;
h=5;
T0=300;
E=500;
Cd=0.6;
cp=1010;
p=1.293;
g=9.8;
pi=3.14;
```

```
B=E*g/ (cp*p*T0);

while e< 0.8
 i=i+1;
 e=e+0.1;
 y1(i)=0.042318/Cd*(e^5/(1-e)).^(1/2);
 y2(i)=(Cd*y1(i))^(2/3)*(2*B*(1-e))^(1/3)*h^(5/3);
end

i=0;
j=0;
while h1 < (h-0.2)
 h1=h1+0.1;
 i=i+1;
  dT(i)=(E/(p*cp*Cd*A2))^(2/3)*(T0/(2*g*(h-h1)))^(1/3);
end

e=0.2:0.1:1;
plot (y1,e);
xlabel ('A/h2');
ylabel('hc/h');

figure (2)
e=0.2:0.1:1;
plot (y1,y2);
xlabel ('A/h2');
ylabel ('V2');

e=0.1:0.1:h1
plot (e,dT);
xlabel ('hc');
ylabel ('dT');

dT=dT';
```

APPENDIX C: Maple-V Listing

```
> h1:=0;
> e:=0.1;
> A1:=1;
> A2:=1;
```

```
> h:=5;
> T0:=300;
> E:=500;
> Cd:=0.6;
> Cp:=1010;
> p:=1.293;
> g:=9.8;
> pi:=3.14;
> B:=E*g/(Cp*p*T0);
> for i from 1to 7 by 1 do e:=e+0.1; y1(i):=0.042318/Cd*(e^5/(1-e))^(1/2);
   y2(i):=(Cd*y1(i))^(2/3)*(2*B*(1-e))^(1/3)*h^(5/3); od;
> for j from 1 by 1 while h1 <(h-0.2) do h1:=h1+0.1;
   dT(j):=(E/(p*Cp*Cd*A2))^(2/3)*(T0/(2*g*(h-h1)))^(1/3); od;
>
>plot(0.042318/Cd*(k^5/(1-k))^(1/2), k=0.2..0.8);
>plot((E/(p*Cp*Cd*A2))^(2/3)*(T0/(2*g*(h-k)))^(1/3), k=0..4.8);
>
```

NOMENCLATURE

A	effective opening area of the compartment, m^2
B	buoyancy flux, m^4s^{-4}
C	constant used in plume flow rate calculation
C_c	contraction coefficient
C_d	discharge coefficient
C_p	specific heat capacity of air, $Jkg^{-1}K^{-1}$
g	acceleration due to gravity, 9.8 ms^{-2}
H	height of compartment, m
h	vertical distance between upper and lower openings, m
h_1	height of smoke zone, m
h_c	height of neutral zone interface, m
L	length of compartment, m
P	perimeter of fire, m
p	air pressure, Pa
Q	heat release rate, W
T	absolute temperature, K
T_0	absolute temperature of ambient air, K
V	volume flow rate, m^3s^{-1}
v	air velocity, ms^{-1}
W	width of compartment, m

α_c convective heat transfer coefficient at the ceiling
α_f convective heat transfer coefficient at the floor
α_r radiative heat transfer coefficient at the ceiling
ρ air density
ρ_0 ambient air density
λ temperature coefficient
ξ resistance coefficient
γ vertical temperature gradient
Δ difference
ΔT difference in temperature

Subscripts

$_1$ inflow
$_2$ outflow
$_c$ contracted
$_d$ discharge

REFERENCES

1. D. Etheridge and M. Sandberg, *Building Ventilation: Theory and Measurement,* John Wiley & Sons, Chichester, 1990
2. W. D. Baines, J. S. Turner, and I. H. Campbell, Turbulent Fountains in an Open Chamber, *Journal of Fluid Mechanics, 212,* pp. 557-592, 1990.
3. P. F. Linden, Emptying Filling Boxes: The Fluid Mechanics of Natural Ventilation, *Journal of Fluid Mechanics, 212,* pp. 309-335, 1990.
4. K. T. Andersen, Theoretical Considerations on Natural Ventilation by Thermal Buoyancy, *ASHRAE Transactions, 101*:2, pp. 1103-1117, 1995.
5. G. M. J. Davies and P. F. Linden, The Effects of Headwind on Buoyancy-Driven Flow through a Doorway, *3rd International Conference on Air Distribution in Rooms, ROOMVENT '92, 3,* pp. 419-433, 1992.
6. P. Cooper and P. F. Linden, Natural Ventilation of an Enclosure Containing Two Buoyancy Sources, *Journal of Fluid Mechanics, 311,* pp. 153-176, 1996.
7. P. F. Linden and P. Cooper, Multiple Sources of Buoyancy in a Naturally Ventilated Enclosure, *Journal of Fluid Mechanics, 311,* pp. 177-192, 1996.
8. W. K. Chow, Assessment of Thermal Environment in an Atrium with Air-Conditioning, *Journal of Environmental Systems, 25*:4, pp. 409-420, 1997.
9. Y. Li, Buoyancy-Driven Natural Ventilation in a Thermally Stratified One-Zone Building, *Building and Environment, 35,* pp. 207-214, 2000.
10. G. Cox, *Combustion Fundamentals of Fire,* Academic Press, New York, 1995.
11. *MATLAB Version 5 User's Manual,* The MathWorks Inc., Massachusetts, 1997.
12. *MAPLE V Version 5.1 User's Manual,* Waterloo Maple Inc., Ontario, Canada, 1998.
13. *PHOENICS Version 3.2 User's Manual,* Concentration, Heat & Momentum Limited, London, United Kingdom, 1998.

14. H. Rouse, Gravitational Convection from a Boundary Source, *Tellus, 4*:3, pp. 201-210, 1954.
15. C. S. Yih, *Fluid Mechanics: A Concise Introduction to the Theory,* West River Press, Ann Arbor, Michigan, 1977.

Direct reprint requests to:

W. K. Chow
Department of Building Services Engineering
The Hong Kong Polytechnic University
Hong Kong, China
e-mail: bewkchow@polyu.edu.hk