

ZONAL MODEL – A SIMPLIFIED MULTIFLOW ELEMENT MODEL

F. Haghghat¹, Y. Lin¹ and A. C. Megri¹

¹Department of Building, Civil and Environment Engineering, Concordia University, Canada

ABSTRACT

Several models have been developed to study the airflow pattern and thermal distribution in buildings. This paper describes the development of a zonal model that may be incorporated into existing building energy analysis programs. The modeling is detailed in three applications describing how such a model can be simply applied. The model predictions were then compared with measurements and/or Computational Fluid Dynamics (CFD) model prediction.

1. INTRODUCTION

Computer simulation model can help to study the thermal comfort and indoor air quality in a room or in a building as well as provide information on the impact of indoor air parameters on these indices. These models can be grouped into two categories: single flow element models that are known as *multi-zone air flow models*, and multi-flow elements models that are known as *CFD models*. Multi-zone modeling approach takes a room within a building as one node that is connected to others by openings between rooms and openings to the outside [1]. The nodes are connected to each other through a line that is called as element. Uniform and instant mixing is assumed for each zone. This approach has the advantage of user friendliness in terms of problem definition; straightforward internal representation and calculation procedure; and easily incorporating into a mass flow balance approach. Their advantages allow the prediction of bulk flows through the whole building as caused by wind, temperature differences, and/or mechanical systems. However, it does not provide information about temperature and airflow distribution within a room due to its simplified approach.

CFD models are capable of predicting two- or three-dimensional airflow and thermal distribution patterns in a room [2]. The governing equations are the continuity, momentum, and energy and mass diffusion equations. Despite of the richness of the results in terms of information detail regarding the flow and temperature field within a room, it suffers from the huge user effort in terms of problem definition and computation efforts. Thus, it is practically difficult to apply this approach by integrating this method within general building energy simulation.

This paper describes the development of a simplified multi-flow element model: it is known as zonal model. The term simplified, as used here, describes a model that provides some global indication of airflow and temperature distributions in a room, as well as a model that is relatively easy for users to define the problem, and is easily incorporated into building energy analysis programs.

2. ZONAL MODEL

In this approach, a room is divided into n well-mixed iso-thermal zones in which parameters such as temperature and contaminants concentration are assumed to be uniform. The air is

also assumed inviscid. There are inter-zonal mass flow and thermal flow between zones. Generally, the thermal and mass balance equation can be written for each zone as:

$$\frac{dM_i}{dt} = \sum_{j=1}^n m_{ij} + m_{source} + m_{sink} \quad (1)$$

$$\frac{dQ_i}{dt} = \sum_{j=1}^n q_{ij} + q_{source} + q_{sink} \quad (2)$$

Where, M_i is mass in the zone i , and m_{ij} is the rate of mass flow from zone i to zone j , m_{source} is the rate of mass supplied by the source in the zone, and m_{sink} is the rate of mass removed from the zone. Q_i is the heat energy in zone i , and q_{ij} is the rate of heat energy from zone i to zone j , q_{source} is the rate of heat energy supplied by the source in zone, q_{sink} is the rate of heat energy removed from the zone.

There is an independent reference pressure at the bottom of each zone. The air pressure in the zone is assumed to be hydrostatically distributed based on this reference pressure. Three kinds of boundaries are identified in the room, i.e. 1). Normal boundary: the boundary without the influence of a jet, 2). Jet boundary: the boundary totally within the plume of a jet, 3). Combined boundary: the boundary on the border of a plume.

2.1. Modeling of the mass flow across normal boundary

The mass flow rate (m) across both the horizontal and vertical boundaries are modeled based on the power law. For the horizontal boundary, the mass flow rate is represented as:

$$m = \rho k (P_{up_ref} - (P_{down_ref} - \rho_{down} g h_{down}))^n A \quad (3)$$

Where ρ is the density of the zone from which the air flows into, k and n are the power law coefficients, and A is the opening area. The subscripts *up*, *down* and *ref* mean the upper zone, the lower zone, and the reference pressure, respectively.

As to vertical boundary, referring to Figure 1, the airflow rate across the vertical boundary can be modeled as:

$$m_{0-Z_n} = kL\rho|\Delta\rho g| \frac{|Z_n|^{n+1}}{n+1} \quad (4)$$

$$m_{Z_n-H} = kL\rho|\Delta\rho g| \frac{|Z_n - H|^{n+1}}{n+1} \quad (5)$$

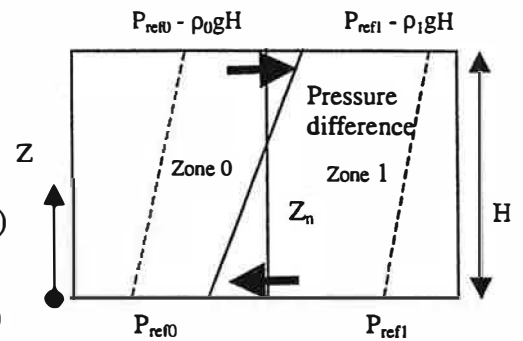


Figure 1. Pressure difference at vertical boundary

where Z_n is the neutral plane, where the pressure

difference across the boundary is zero, $Z_n = \frac{\Delta P_{ref}}{\Delta \rho g}$, ΔP_{ref} is the reference pressure difference,

$\Delta \rho$ is the density difference between the two adjacent zones, k and n have the same meanings as those in equation (3), and L is the depth of the zone. m_{0-Z_n} is the rate of mass flow from 0 to Z_n in the vertical boundary, m_{Z_n-H} is the rate of mass flow from Z_n to H , ρ is the density of zone from which the air flows into. This ρ can be calculated by means of ideal gas law.

Explicitly, there are 8 possibilities of airflow pattern across the vertical boundaries, depending on whether Δp is larger than zero and on the position of the neutral plane Z_n .

2.2. Modeling of the mass flow across jet boundary

The mass flow across the jet boundary vertical to the trajectory of the jet can be modeled as [3]:

$$m = \int_0^m dm = \int_a^b \rho V_x 10^{-\frac{1}{3.3} \left(\frac{r}{x \tan \alpha / 2} \right)^2} L dr \quad (6)$$

where, a and b are lower and upper coordinates from the center line of the jet, ρ is the density of the flow, and V_x is the centerline velocity, which can be calculated based on equations in ASHARE Fundamentals [3]. x is the distance from the boundary to the inlet, α is the jet expansion angle, usually taken as 22° [1], and L is the deep of the jet and considered as unit in 2D configuration.

2.3. Modeling of the mass flow across the combined boundary

On the combined boundary, i.e. the boundary in Figure 2, the partition (a-H) is within the jet and the partition (0-a) is out of the jet. The airflow from a to H (m_{a-H}) can be modeled by applying equation (6). As to the airflow from 0 to a (m_{0-a}), it can be modeled as the airflow across the normal boundary.

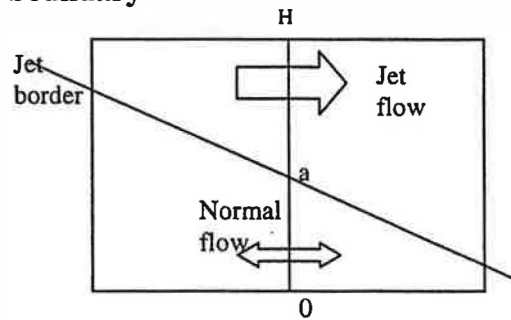


Figure 2. Combined boundary

$$m_{0-a} = \frac{a}{H} * m_{0-H}^{normal} \quad (7)$$

Where, m_{0-H}^{normal} is the airflow across the normal boundary (see section 2.1). Consequently, the total airflow across the mixed boundary, m_{0-H} , is the sum of m_{0-a} and m_{a-H} .

2.4. Modeling of the heat flow

Ignoring the radiation component, the heat flow is a combination of convective heat transfer and the heat carried by mass flow. The convection is modeled by means of the Newton Cooling Law, $q = hA\Delta T_h$, and the heat carried by the mass flow can be modeled as $q = C_p m \Delta T_m$, in which q is the heat flow rate, h is the convective heat transfer coefficient, A is the surface area, C_p is the heat capacity of the air, m is the mass flow rate, ΔT_h is the temperature difference between wall surface and the air of the zone, and ΔT_m is the temperature difference in the two zones.

It is noted that there are only $(n-1)$ independent mass balance equations in a system. It can be easily demonstrated for a room with 4 zones and it can be extended to a room with n zones by using linear algorithm. The unknown factors are n temperatures and n pressures in n zones. Since we are only concerned with pressure difference, let us assume a basic reference pressure

for all zones so that there are (n-1) pressure differences for n pressures. Hence the total number of unknowns is (2n-1), which is also the number of independent equations.

3. SIMULATION RESULTS AND VALIDATION:

The prediction of the Pressurized zonal Model with Air-diffuser (POMA) is compared with the experimental results and with the prediction of another zonal model as well as CFD model. The comparisons are made for both natural and forced ventilation cases.

3.1. Natural Convection

The first case study is a window problem for stratification prediction in a steady state condition. The inside surface temperature and other data are listed in Table 1. The room is simulated with a 4x1x4 uniform mesh. The results given by our zonal model (Figure 3), agree well with those given by the CFD model FLOVENT (36x1x36 uniform mesh), as shown in Figure 4. A large circulation along the wall is detected in both figures. As well, the similar airflow patterns are shown in these two figures.

Table 1. Input data of window problem

	Length (m)	Surface temp	H(w/m ² C)
Top	6.0	15	5.7
Bottom	6.0	15	1.0
Left	2.4	12	4.2
Right	2.4	20	4.2

Table 2 Inside surface temperature (°C) of four cases of natural convection for MINSBAT test cell

	South	North	East	West	Ceiling	Floor
Case 1	6.0	13.9	14.1	14.1	13.5	11.8
Case 2	16.9	33.0	26.9	27.3	28.5	25.9
Case 3	15.3	29.1	26.1	26.2	26.0	27.6
Case 4	11.2	23.8	23.5	23.7	42.1	21.1

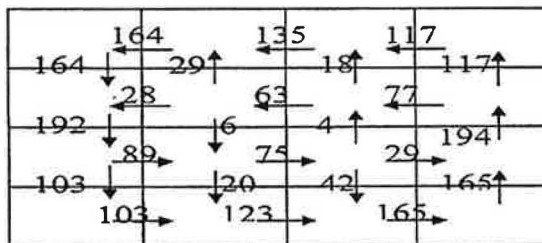


Fig-3. Airflow (kg/h) and temperature (°C) predicted by POMA with 4x4

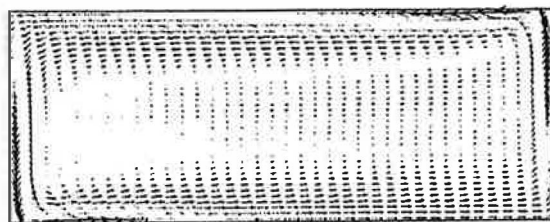
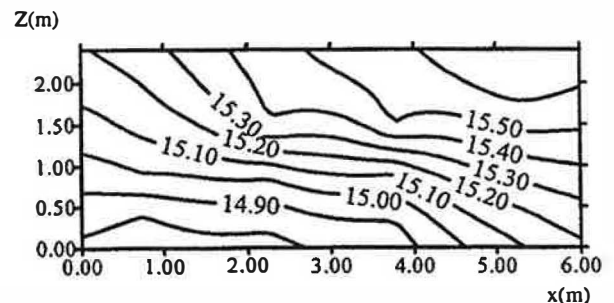
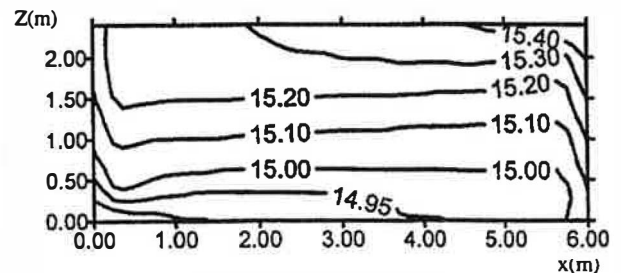


Fig-4. Airflow and temperature (°C) predicted by FLOVENT with 36x36 zones



3.2. MINIBAT Test Cell

A MINIBAT test cell (3.1×3.1×2.5m) in CETHIL (Centre de Thermique de l'INSA de Lyon) is used to collect data for four cases of natural convection in a steady state condition [4]. The measured values of inside surface temperature are list in Table 2. Without taking into account radiation, the test cell is simulated with a 2D mesh (6×1×10 zones). The temperature distribution results are given in Figure 5. From the figures, we can see the simulation results agree well with the experimental results and Inard's results. Compared to Inard's results, the simulation results can reflect the temperature gradient near the walls.

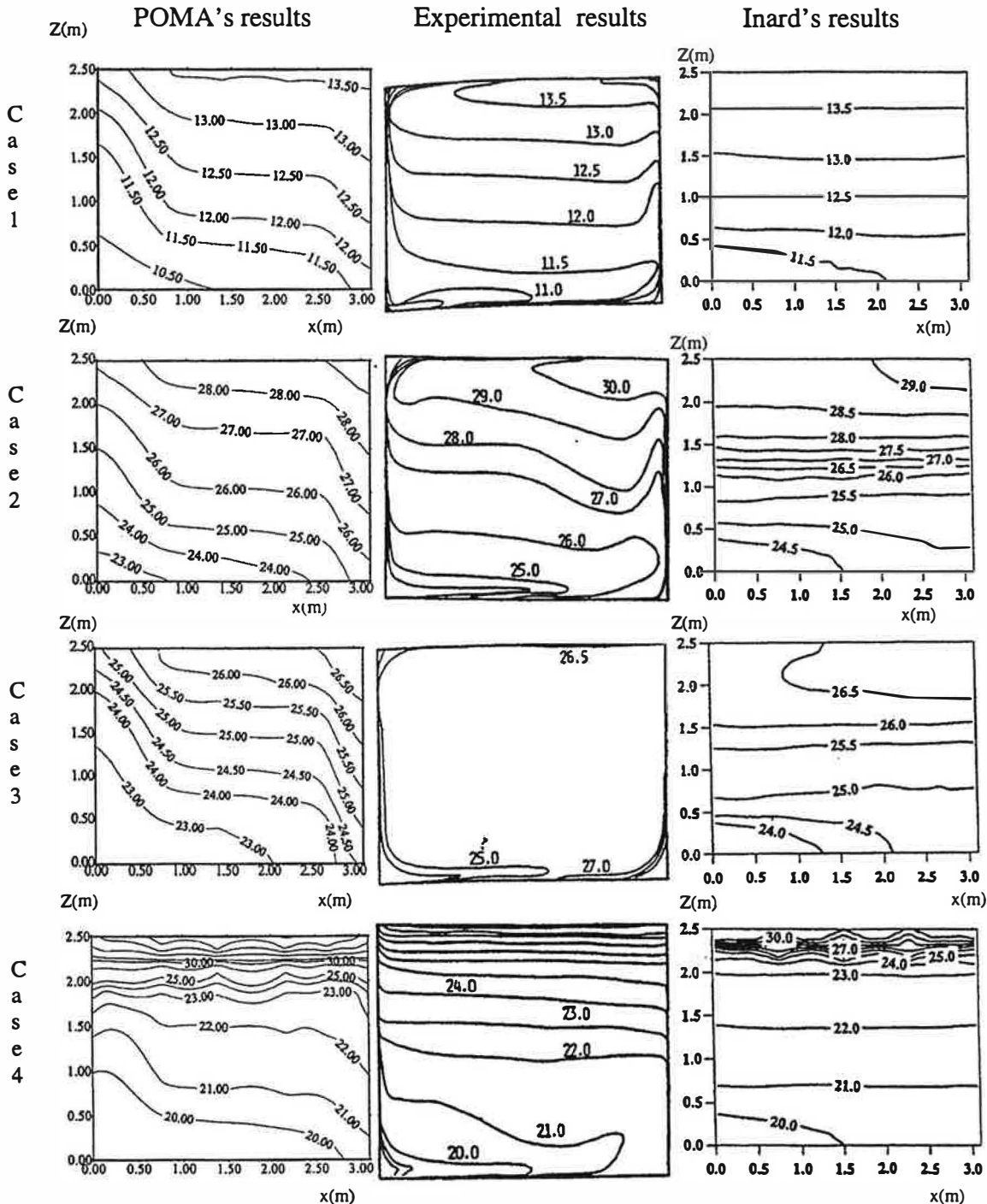


Figure 5. Temperature (°C) distribution of four cases of natural convection

3.3. Cross ventilation

The cross ventilation case is a 3×3×3m room. There are two openings 0.5m high on opposite walls. All of the wall surface temperatures are 18°C. The inlet air velocity is 0.5m/s. The room is simulated with 2D meshing (3×1×5) in two situations, i.e. 1). with isothermal jet and 2). with nonisothermal jet. In isothermal situation the inlet air temperature is 18°C, while in non-isothermal situation the inlet air temperature is 20°C. In isothermal situation, the airflow pattern simulated by CFD is shown in Figure 6 [3]. Comparing the airflow pattern with that simulated by the zonal model, shown in Figure 7, we can see that the flow patterns are the same. Recirculating flows near the top and bottom walls are detected and the airflow patterns are symmetric in the two figures. Moreover, these figures also show that there is some agreement between the prediction of the flow volumes.

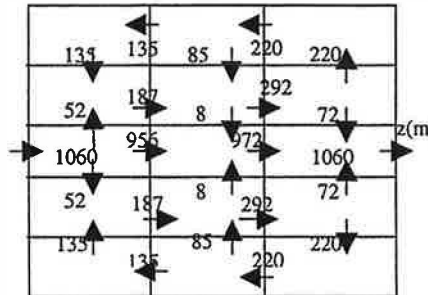


Figure 6. Airflow (kg/h) in isothermal situation predicted by CFD

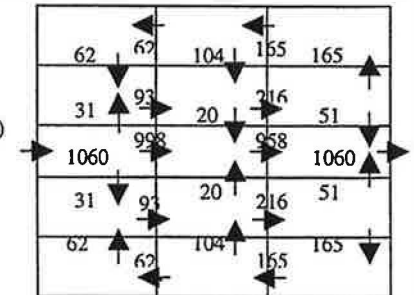


Figure 7. Airflow (kg/h) in isothermal situation predicted by POMA

Figure 8 and Figure 9 show the results of the zonal model for the room in non-isothermal situation. The recirculation of the flow is detected. The flow pattern and temperature distribution are detected to be slightly asymmetrical due to the thermal buoyancy. It demonstrates that this zonal model has the capability of predicting the influence of thermal buoyancy.

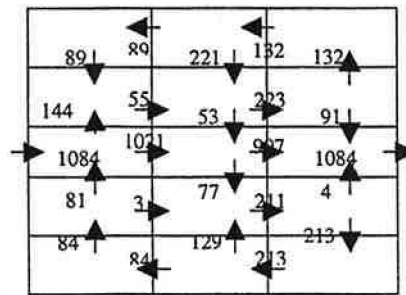


Figure 8. Airflow (kg/h) in non-isothermal situation predicted by POMA

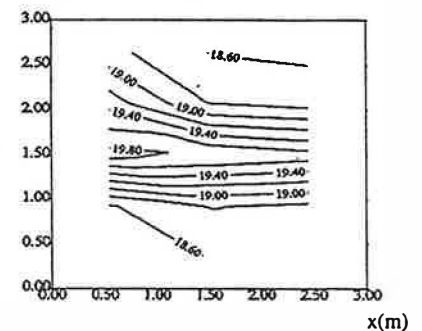


Figure 9. Temperature (°C) distribution in non-isothermal situation predicted by POMA

4. CONCLUSION

A simplified zonal model to predict the airflow pattern and temperature distribution in a room is developed and validated with existing information from literature. The agreement between the simulation and experimental result demonstrated that the zonal model is a feasible approach of thermal simulation in the view of engineering. Its simple use gives it advantages to be integrated into existing multi-room building energy and airflow analysis programs, such as COMIS or CONTAM [1]. This model will be used to predict the effect of radiative and convective heating and cooling systems on thermal comfort, in order to accurately provide design information and properly size radiative and convective systems, as well as hybrid systems composed of forced-air convective components.

5. ACKNOWLEDGEMENT

This work is part of the Canadian contribution to the work of Annex 35 of the International Energy Agency "Hybrid Ventilation in New and Retrofitted Office Buildings". Funding for the first author participation at the meeting was made possible through a contract by the Energy Efficiency Division of Natural Resources Canada.

6. REFERENCE

1. Haghighat, F. and Megri, A.C. (1996) " A Comprehensive Validation of two Airflow Models: COMIS and CONTAM", *INDOOR AIR*, vol. 6, pp. 278-288.
2. Haghighat, F., Jiang, Z., Wang, J.C.Y. and Allard, F. (1992) "Air Movement in Buildings Using Computational Fluid Dynamics", *The ASME Journal of Solar Energy Engineering*, vol. 114, No. 2, pp. 84-92.
3. ASHRAE Fundamentals 1993. *The American Society of Heating, Refrigerating and Air-Conditioning Engineers*, Chapter 31.
4. Inard, C, Bouia, H and Dalicieux, P 1996. Prediction of air temperature distribution in building with a zonal model, *Energy & Buildings*, Vol. 24, pp 125-132
5. Rodriguez, E and Allard, F 1995. Zonal modeling for natural ventilation, *Final report of ventilation thermal mass subtask in PASCOOL, chapter 7*