



SIMPLIFIED APPROACH TO EVALUATE INDOOR AIR TEMPERATURE DISTRIBUTION

Admilson. T. Franco

Laboratory of Thermal Science, Academic Department of Mechanics, Federal Centre of Technological Education of Paraná, Rua Sete de Setembro, 3165 - CEP-80230-901 Curitiba/PR – Brazil - e-mail: admilson@cefetpr.br

Cezar. O. R. Negrão

Laboratory of Thermal Science, Academic Department of Mechanics, Federal Centre of Technological Education of Paraná, Rua Sete de Setembro, 3165 - CEP-80230-901 Curitiba/PR – Brazil - e-mail: negrao@cefetpr.br

***Abstract.** This paper describes a simplified computational model to predict indoor air temperature distribution. The model is based on the energy conservation equation combined with a scaling analysis of the momentum equation. Two or three-dimensional domains can be discretized in a number of finite volumes and the energy balance is considered for each volume. The resulting set of non-linear equations is iteratively solved using the line-by-line Thomas Algorithm. As long as the only equation to be solved is the conservation of energy and its coefficients are not strongly dependent on the temperature field, the solution is considerably fast. Therefore, the application of such model to a whole building system is quite reasonable. Case studies were carried out and comparisons with CFD solutions were performed. The results are quite promising for engineering purposes.*

***Key-words:** Simplified model, Displacement ventilation, CFD.*

1. INTRODUCTION

The common practice in building thermal analysis is to assume mixed flow within spaces (Clarke, 1985). In other words, all air properties (such as, temperature, humidity, etc) are considered uniform inside zones. This hypothesis is valid when the interest is on the thermal behaviour of a whole building. Although uniform zone air might be an acceptable assumption for many problems where the focus is on the long-term energy matters, this simplification is not valid for cases involving relatively strong couplings between heat and airflow or relatively high temperature gradients. Displacement ventilation is a typical example of such case provided the flow is mostly induced by natural convection.

On the other hand, Computational Fluid Dynamics (CFD) has emerged as a robust tool for analysis of complex flows (Patankar, 1980). In building cases, CFD allows computation of indoor air temperature distribution and air velocity gradients. However, the number of equations to represent a multi-zone problem is considerably high and their solution is complicated. Besides, the characteristics of indoor air motion are always difficult to identify whether it is locally induced, transitional or fully turbulent. This fact introduces an additional complexity into the modelling which is not completely resolved for building applications.

Negrão (1998) has proposed an approach to combine the CFD and the mixed air model. However, a whole year simulation applied to multi-store buildings is still prohibited.

In the current work, a simplified computational model is presented in order to predict indoor air temperature distribution. Two or three-dimensional domains can be discretized in a number of finite volumes and the energy balance is applied to each volume. Air velocity is estimated by a scaling analysis and the computed airflow is imposed in certain volumes.

Vargas et al. (2001) presented this approach, and they called it volume element model (VEM). They have solved the air temperature and humidity within an electronic cabinet and validated the results with experimental data. However, their model considers each finite volume as a Bernard cell, which is specific for their case study. In the present study, the energy balances are established according to the geometry and boundary conditions, which is again valid for the current case.

In this work, the VEM is applied to a displacement ventilation case. The results are discussed and compared with CFD ones.

2. MATHEMATICAL MODEL

The VEM modelling application is much dependent on the problem under consideration. The developer/user must have a fair idea of the geometry and boundary conditions. Therefore, a case study was firstly established and the domain geometry is shown in Fig. (1). In order to demonstrate the approach, the case is considered two-dimensional a heat source of 950W is placed in the midway of the room width and all surface temperatures are set to 22°C. The heat source, however, does not obstruct the passage of the air. It is just a source of heat imposed in the air volume.

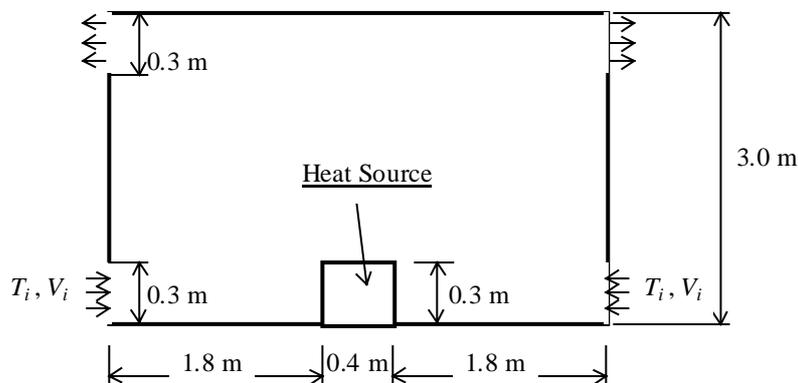


Figure 1 – Case study geometry.

Air is supplied to the cavity by floor level openings (0.3m high) with temperature of $T_i = 20^\circ\text{C}$ and speed of $V_i = 0.2\text{ m/s}$. One must note that a plume rises above the source as a result of air heating. Considering the problem symmetry, the generated flow must be diverted equally to both sides when the top wall is reached. The incoming flow leaves the cavity through the top openings (0.3m high) while the remaining flow forms two descending streams near the vertical walls. The recirculating streams will mostly flow in a region close to the walls within the boundary layer, as shown in Fig. (2). Four regions will thus be considered in the modelling: three fluid flow regions (near the inlet and outlets, at and above the heat source and close to the walls) and a heat diffusion region far from the walls.

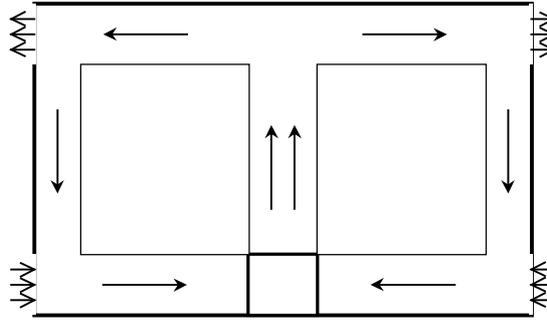


Figure 2 – Schematic representation of fluid flow.

After those considerations, the whole domain is divided in a number of finite volumes (cells) and an energy balance equation is written for each volume. Four types of cells will be considered, according to the region they are placed. Those equations are presented next for each region.

Near the bottom at the inlet.

Consider the cell P close to the bottom left entrance, shown in Fig. (3a). Applying an energy balance to this cell:

$$\dot{m}_h c_{pa} T_H + h_l A_l (T_l - T_P) - \dot{m}_n c_{pa} T_P + \dot{m}_i c_{pa} T_i = 0 \quad (1)$$

where, T is the temperature, h is the convection coefficient, A is the area, c_{pa} is the specific heat of the air and \dot{m} is the mass flow rate. The subscript H represent the upper neighbouring cell of P and the subscripts h , l and n refers, respectively, to the interface between P and H , L , and N cells or the corresponding boundary. The subscript i refers to the inlet. A similar balance can be applied to the other inlet and to the outlets.

Near the wall region.

Consider the cell P near the left vertical wall, shown in Fig. (3b). A steady energy balance applied to this cell results in the following equation:

$$\dot{m}_h c_{pa} T_H - \dot{m}_l c_{pa} T_P + h_s A_s (T_s - T_P) + \frac{k_n A_n}{\Delta x_n} (T_N - T_P) = 0 \quad (2)$$

where Δx is the distance between two cells and k is the air conductivity. The subscripts N represent the right neighbouring cell of P and the subscript n refers to the interface between P and N cells or the corresponding boundary. One should note that the first and second terms in Eq. (2) represent advection, the third represents convection at the wall and the last one, diffusion of heat. Equation (2) can be applied to any cell close to the wall, however, the direction of flow must be taken into account.

Heat source region

Applying the energy conservation equation to cell P in Fig. (3c), the following equation is obtained:

$$\dot{m}_l c_{pa} T_L - \dot{m}_p c_{pa} T_P + \frac{k_s A_s}{\Delta x_s} (T_s - T_P) + \frac{k_n A_n}{\Delta x_n} (T_N - T_P) = 0 \quad (3)$$

Note that advection takes place at vertical direction and diffusion in the horizontal direction.

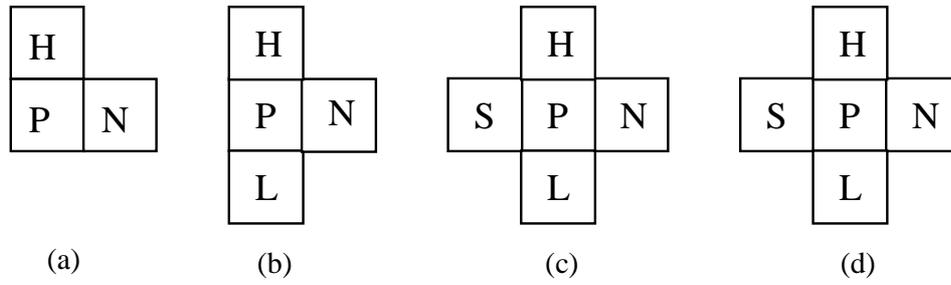


Figure 3 - Dcretization scheme. Cell P and its neighbours: (a) at the inlet supply, (b) close to the wall, (c) at/above heat source and (d) in the middle of the domain.

Middle of the room region

The energy conservation applied to the middle cells in Fig. (3d) results in the equation:

$$\frac{k_l A_l}{\Delta z_l} (T_L - T_p) + \frac{k_h A_h}{\Delta z_h} (T_H - T_p) + \frac{k_s A_s}{\Delta x_s} (T_S - T_p) + \frac{k_n A_n}{\Delta x_n} (T_N - T_p) = 0 \quad (4)$$

In this part of the domain, only diffusion of heat is considered to take place.

Convection coefficients

The convection heat transfer coefficients can be obtained from the literature (Alamdari and Hammond, 1983) and these are usually dependent on the air temperature and flow characteristics:

$$h = \left\{ \left[a \left(\frac{\Delta T}{d} \right)^p \right]^m + \left[b (\Delta T)^q \right]^m \right\}^{1/m} \quad (5)$$

where a , b , p , q and m are given in Tab. (1).

For horizontal surfaces undergoing downward heat flow the natural convective heat transfer coefficient is given by:

$$h = 0.6 \left(\frac{\Delta T}{d} \right)^{1/5} \quad (6)$$

where ΔT is the difference of temperature between the air and the surface. For vertical surfaces, the characteristics dimension d is given by the surface height, whereas for horizontal surfaces the characteristic dimension is the hydraulic diameter, found from

$$d = \frac{4A}{P} \quad (7)$$

where A is the surface area (m^2) and P is the perimeter length (m).

Table 1 – Empirical coefficients for Eq. (5) from Alamdari and Hammond (1983).

Surface aspect	a	b	p	q	m
Vertical	1.5	1.23	1/4	1/3	6
Horizontal	1.4	1.63	1/4	1/3	6

Mass flow rates

Bejan (1984) has proposed a scale analysis based on momentum, energy and mass conservation equations for natural convection at vertical boundary layers. From his analysis, the order of magnitude of the air velocity can be estimated:

$$v = \frac{\alpha}{H} (RaPr)^{1/2} \quad \text{if } Pr < 1 \quad (8)$$

where Pr is the Prandtl number of air and Ra is the Rayleigh Number:

$$Ra = \frac{g\beta \Delta T H^3}{\alpha \nu}$$

where g is the gravitational constant, β is the expansion coefficient, H is the plate height, α and ν are, respectively, the thermal and hydrodynamic diffusions. The ΔT is the difference between the wall temperature and the air temperature outside the boundary layer.

Equation (8) is employed to estimate the air velocity at the plume. This vertical velocity is considered uniform to all cells at and above the heat source. In the current case, the Rayleigh number is based on the domain height and on the maximum temperature difference within the domain (maximum air temperature at the heat source minus the wall temperature).

At the plume, the incoming flow is added to that generated by the heat source. The mass flow rate at the heat source is computed based on the vertical air velocity and on the area of the cells above the source. The airflow close to the vertical walls is half of that generated by the heat source and circulates at the row of cells closer to the walls.

As the air flow and convection coefficients are dependent on the air temperature and vice versa, an iterative procedure is necessary to solve the set of algebraic equations.

3. METHOD OF SOLUTION

The application of the Eqs. (1) to (4) to all finite volumes in the domain originates a set of non-linear algebraic equations:

$$A_p T_p = A_s T_s + A_n T_n + A_l T_l + A_h T_h + S_p \quad (9)$$

where A 's are the coefficients that depend on the mass flow rate, convection coefficients and air conductivity. The set of algebraic Eqs. (9) is solved by the interactive line-by-line Thomas Algorithm (Patankar, 1980).

4. RESULTS

The geometry of Fig. (1) is divided in 12x11 finite volumes and the temperature distribution is shown in Fig. (4). The results showed to be insensitive for a greater number of finite volumes and the computational time to achieve the converged solution was minimal. The velocity estimated by Eq. (8) is in the order of 0.7m/s. The convection coefficients are computed by Eqs. (5) and (6) and the heat flux is based on the difference of wall temperature and the temperature of the cells nearest to the wall. As expected, the highest temperature in the cavity is within the heat source. Also, the isotherms indicate the imposed flow circulation, the outgoing flow in the upper slots, and cold incoming flow at the lower slots. The temperature is stratified, as supposed to be in displacement ventilation, and the difference of temperature from top to bottom walls is 2°C, as shown in Tab. (2). The average air temperature evaluated by the mixed flow model is 26.0°C, which is not representative of the whole room temperature; temperature varies from 24.2 to 27.1°C.

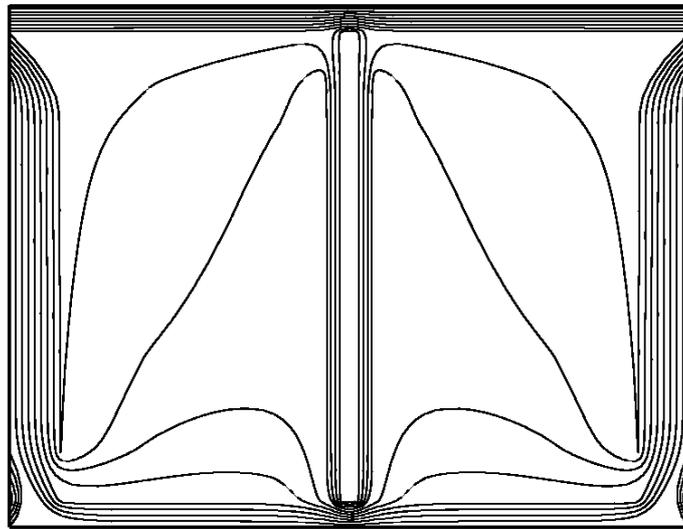


Figure 4 - Temperature distribution inside the cavity.

A sensibility analysis was conducted by changing the wall temperatures to 25°C. Figure (5) shows that the temperature gradient reduce at the walls and increase at the inlets.

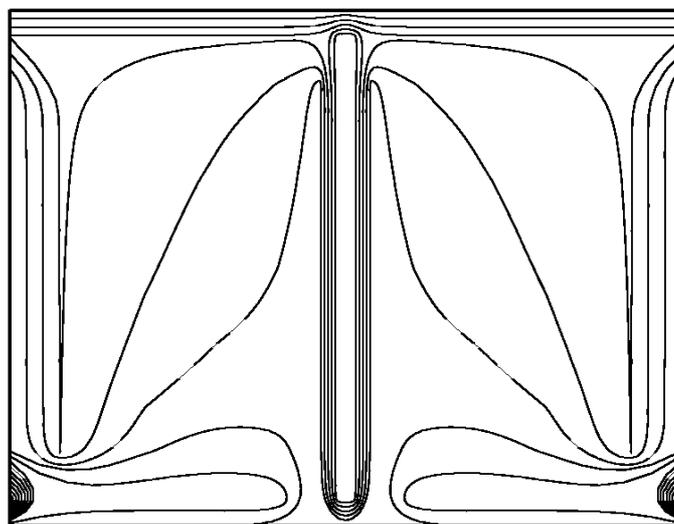


Figure 5 – Temperature distribution for wall temperatures equal to 25°C.

In order to corroborate the results, an inter-model comparison was conducted. A finer grid was employed (30x21) once the results are sensitive to the number of cells at the heat source region. Surface temperatures were changed to 21°C. The CFD modelling was considered and the Flotherm™ (2001) package was employed as the base model. The Flotherm™ model was built for the geometry of Fig. (1).

The Flotherm™ (2001) solves the steady-state Navier-Stokes and energy equations and the turbulent flow is modelled by the Turbulent k-ε Revised Model of Flotherm™, which is an enhancement of the Launder and Spalding (1974) k-ε standard model. The equations are discretized by the Patankar's (1980) SIMPLE method. The employed Cartesian mesh of 192x83 points (15,936 control volumes) is refined where the higher gradients are expected; namely, close to the source, walls and openings. A SUN Ultra Enterprise 450 Workstation (2 SUNW processors, 296MHz, 768Mbytes RAM) was employed in the CFD simulation. The necessary CPU time is in the order of 30 min.

Figure (6) illustrates the Flotherm™ velocity field which states a clear flow close to the floor, ceiling and above the source and a smaller recirculation flow. As foreseen, the velocities in the central region of the room assume low values and definitely only diffusion takes place in that region. One can observe the complexity of the fluid flow inside the room.

With the purpose to compare the isotherms of the models, some information necessary for VEM was obtained directly from the Flotherm™ results. The plume velocity (Fig. (6)) acquired from the Flotherm™ results is in the order of 0.3 m/s. The width/height of the first row of cells close to the wall was also estimated from Fig. (6) (width of cells near the vertical walls = 0.25 m; height of cells near the top and bottom walls = 0.3 m).

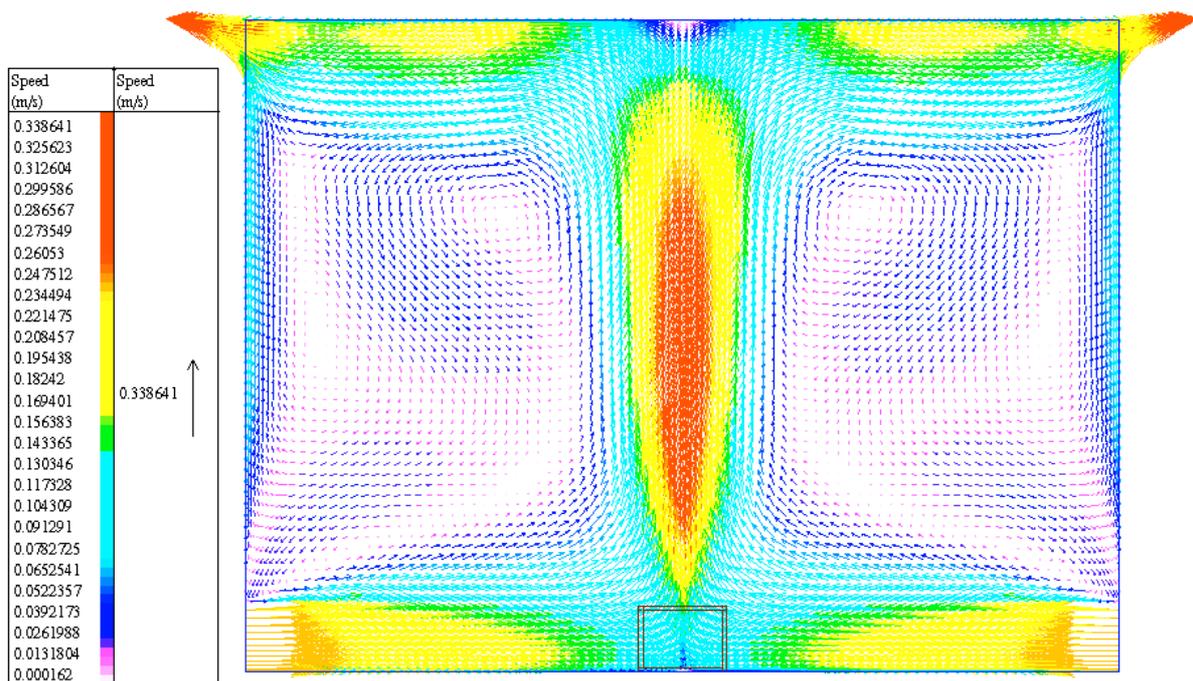


Figure 6 –The velocity field obtained with the k-ε revised turbulence model of Flotherm™.

Figure (7) shows that the isotherms of Flotherm™ are quite similar to those of the current model. The range of temperature within the cavity varies from 21°C to 26°C in the VEM results and from 20.0°C to 25.2°C in the CFD results. Both present a plume above the heat source. Also, the highest temperature gradients are near the heat source (the isotherms are closest to each other). Nevertheless, the largest difference between the profiles is in the plume, which is explained by the different velocities in that region. On one hand, the CFD velocity distribution (Fig. 6) shows that

the plume width increases as the flow rises, and on the other hand, an uniform velocity is imposed in all cells above the heat source. An enhancement on the plume modelling can definitely approximate the CFD and the current model results.

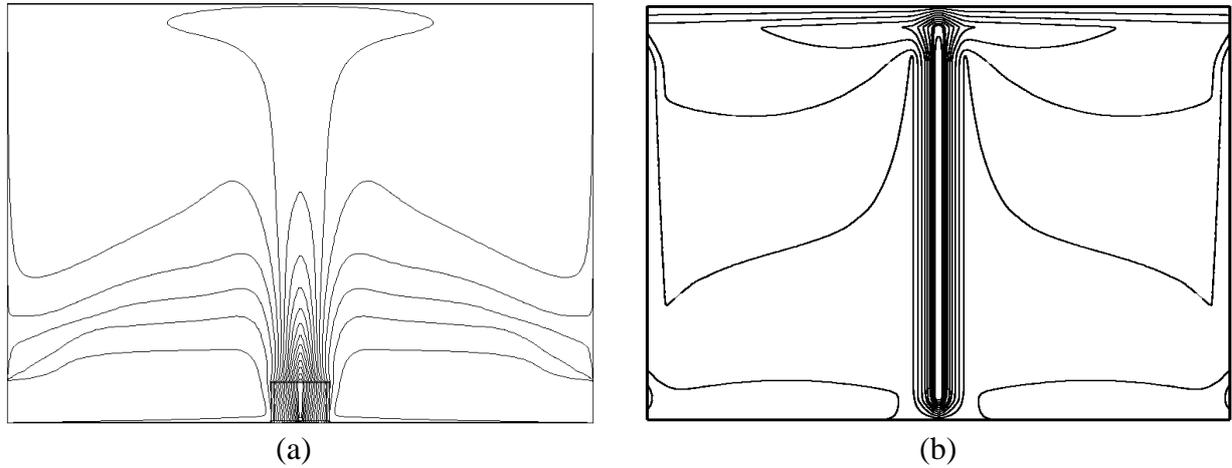


Figure 7 - Comparison of isotherms produced by (a) Flotherm™ and (b) the current model.

Table 2 – Temperature within the cavity in °C

x(m) y(m)	0.125	0.508	1.025	1.542	1.850	1.950	2.050	2.150	2.458	2.975	3.492	3.875
2.850	26.13	26.13	26.14	26.14	26.14	27.12	27.12	26.14	26.14	26.14	26.13	26.13
2.567	26.11	26.05	25.95	25.73	25.17	27.12	27.12	25.17	26.73	25.95	25.05	26.11
2.300	26.10	25.98	25.81	25.53	25.17	27.12	27.12	25.17	25.53	25.81	25.98	26.10
2.033	26.09	25.92	25.69	25.42	25.17	27.12	27.12	25.17	25.42	25.69	25.92	26.09
1.767	26.07	25.85	25.59	25.34	25.17	27.12	27.12	25.17	25.34	25.59	25.85	26.07
1.500	26.06	25.78	25.49	25.28	25.17	27.12	27.12	25.17	25.28	25.49	25.78	26.06
1.233	26.05	25.68	25.36	25.20	25.17	27.12	27.12	25.17	25.20	25.36	25.68	26.05
0.967	26.03	25.54	25.20	25.10	25.17	27.12	27.12	25.17	25.10	25.20	25.54	26.03
0.700	26.02	25.32	24.96	24.94	25.17	27.12	27.12	25.17	24.94	24.96	25.32	26.02
0.433	26.01	24.94	24.64	24.68	25.17	27.12	27.12	25.17	24.68	24.64	24.94	26.01
0.150	24.22	24.21	24.20	24.19	25.17	27.13	27.13	25.17	24.19	24.20	24.21	24.22

5. DISCUSSION AND CONCLUSIONS

The current paper presents a simplified numerical model to predict indoor temperature distribution based on the discretization of the energy equation and on the scale analysis of the momentum conservation equation. A comparison with a CFD model was conducted and the results are quite similar to each other. The current model results - volume element model (VEM) - however, are obtained with much less computational effort and its application to building simulation programs seems to be very promising.

The model is much dependent on the insight of the modeller/user and information of some more refined methods such as CFD or experimental set-ups can be used to create VEM equations.

As long as most of the differences between CFD and the current model lies on the plume region above the heat source, an introduction of a more accurate plume model in the VEM could improve the comparison.

Experimental and inter-model comparisons still need to be done in order to consolidate the approach. Calibration of Eq. (8) is also necessary.

6. REFERENCES

- Alamdari F. and Hammond, G. P., 1983, Improved Data Correlation for Buoyancy-Driven Convection in Rooms, Rep. SME/J/83/01, Cranfield Institute of Technology, Applied Energy Group, Cranfield
- Bejan, A., 1984, Convection Heat Transfer, John Wiley & Sons.
- Clarke, J. A., 1985, Energy Simulation in Building Design, Adam Hilger.
- Flotherm™, Flomerics®, 2001, Online Documentation, Version 3.2.
- Launder, B. E. and Spalding, D. B., 1974, "The Numerical Computation of Turbulent Flow", Computer Methods in Applied Mechanics and Engineering, Vol. 3, pp. 269-289.
- Negrão, C. O. R., 1998, "Integration of Computational Fluid Dynamics with Building Thermal and Mass Flow Simulation", Energy and Buildings, Vol. 27-2, pp. 155-165.
- Patankar, S. V., 1980, Numerical Heat Transfer and Fluid Flow, Taylor and Francis.
- Vargas, J. V. C., 2001, Stanescu, G., Florea, R. and Campos, M. C., "A Numerical Model to Predict the Thermal and Psychrometric Response of Electronic Packages", ASME Journal of Electronic Packaging, Vol. 123(3), pp. 200-210.