Building Simulation
Seventh International IBPSA Conference
Rio de Janeiro, Brazil
August 13-15, 2001

SIMPLIFIED MODEL TO PREDICT INDOOR AIR TEMPERATURE DISTRIBUTION

C. O. R. Negrão(*), A. T. Franco(**), L. M. Macedo(*)
(*) Laboratory of Thermal Science
(**) Concurrent Engineering R & D Group
Federal Centre of Technological Education of Paraná
Academic Department of Mechanics
Rua Sete de Setembro, 3165
CEP-80230-901 Curitiba/PR – Brazil
e-mail: negrao@cefetpr.br, franco@nupes.cefetpr.br

ABSTRACT

This paper describes a simplified computational model to predict indoor air temperature distribution. This model is based on the energy conservation equation combined with a scaling analysis of the momentum equation. 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.

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 simplifying approach is presented in order to predict indoor air temperature distribution. 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.

This approach is based on a methodology presented by Vargas et al. (2001), called 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 considered 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.

MATHEMATICAL MODEL

The application of the modelling is much dependent on the problem under consideration. The developer/user must have a fair insight of the geometry and boundary conditions. Therefore, a case study was firstly established and the domain geometry is shown in Figure 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 25^{o}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.

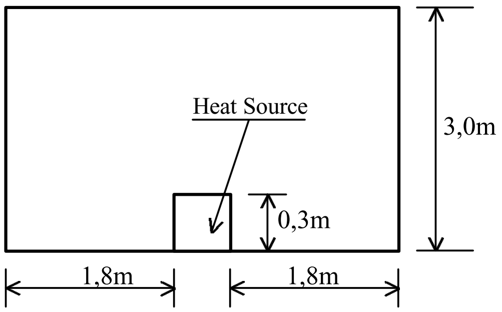


Figure 1 – Case study geometry.

One must note that a plume is generated above the source as a result of air heating. Considering the symmetry of the problem, the generated flow must be diverted equally to both sides when the top wall is reached. As the two streams cool down at vertical walls, the flow must descend close to those walls. The air will mostly flow in a region close to the walls within the boundary layer, as shown in Figure 2. Three regions will thus be considered in the modelling: a fluid flow region at and above the heat source, a fluid flow region close to the walls and a heat diffusion region far from the wall.

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

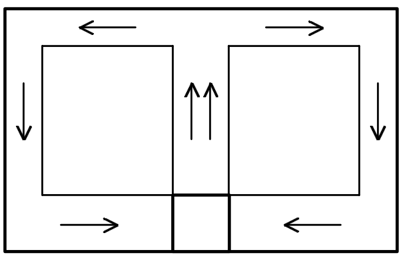


Figure 2 – Schematic representation of fluid flow.

Near the wall region.

Consider the cell P close to the left vertical wall, shown in Figure 3a. A steady energy balance applied to this cell results in the following equation:

$$\dot{m}_h c_{pa} T_H - \dot{m}_l c_{pa} T_P + h_s A_s (T_s - T_P) + \frac{k_n A_n}{\Delta x_n}(T_N - T_P) = 0 \quad (1)$$

Where, T is the temperature, h is the convection coefficient, A is the area, k is the air heat conduction, c_{pa} is the specific heat of the air and Δx is the distance between two cells. The subscripts H and N represent, respectively, the upper and the right neighbouring cells of P and the subscripts h, l, s and n refers, respectively, to the interface between P and H, L, S and N cells.

One should note that the first and second terms in equation (1) represent advection, the third represents convection at the wall and the last one, diffusion of heat. Equation (1) 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 Figure 3b, the following equation is obtained:

$$\dot{m}_l c_{pa} T_L - \dot{m}_P c_{pa} T_P + \frac{k_s A_s}{\Delta x_s}(T_S - T_P) + \frac{k_n A_n}{\Delta x_n}(T_N - T_P) = 0 \quad (2)$$

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

Middle of the room region

The energy conservation applied to the middle cells in Figure 3c results in the equation:

$$\frac{k_l A_l}{\Delta z_l}(T_L - T_P) + \frac{k_h A_h}{\Delta z_h}(T_H - T_P) \\ \frac{k_s A_s}{\Delta x_s}(T_S - T_P) + \frac{k_n A_n}{\Delta x_n}(T_N - T_P) = 0 \quad (3)$$

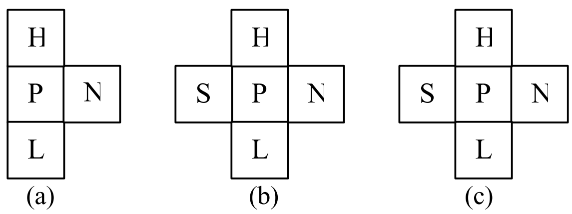


Figure 3 - Dicretization scheme. Cell P and its neighbours: (a) close to the wall, (b) at/above heat source, and (c) in the middle of the domain.

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 (Clarke, 1985 and

Alamdari and Hammond, 1983) and these are usually dependent on the air temperature and flow characteristics.

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:

$$v = \frac{\alpha}{H}\left(RaPr\right)^{1/2} \qquad \text{if} \quad Pr < 1 \qquad (4)$$

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

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

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 (4) 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. 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).

The mass flow rate is computed based on the vertical air velocity and on the area of the cells above the source. The airflow close to the horizontal and vertical walls is half of the flow generated at the plume and this 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.

METHOD OF SOLUTION

The application of the equations (1) to (3) to all finite volumes in the domain originates a set of non-linear algebraic equations:

$$A_P T_P = A_s T_S + A_n T_N + A_l T_L + A_h T_H + S_P \qquad (5)$$

Where A´s are the coefficients that depend on the mass flow rate, convection coefficients and air conductivity. The set of algebraic equations (5) is solved by the interactive line-by-line Thomas Algorithm (Patankar, 1980).

RESULTS

The geometry of Figure 1 is divided in 11x12 finite volumes and the temperature distribution is shown in Figure 4. The computational time to achieve the converged solution was minimal. The velocity estimated by equation (4) is in the order of 0.7m/s. A constant value of convection coefficients was established at the walls (8W/m^{2}K) and the heat flux is based on the difference of wall temperature and the temperature of the cell nearest to the wall. As expected, the highest temperature in the cavity is within the heat source. Also, the isotherms indicate the imposed circulation of flow. The heat flows at the walls are 201.9W, 201.9W, 254.0W, 292.3W, respectively, at the south, north, low and high walls. The temperature is stratified and the difference of temperature from top to bottom walls reaches 1.7^{o}C, as shown in Table 1. Based on the above heat flows and on the convection coefficients, an average air temperature of 33.5^{o}C can be computed. This is the result of the mixed flow model which is not representative of the whole room temperature, as the air it changes from 32.6 to 35.4^{o}C (Table 1).

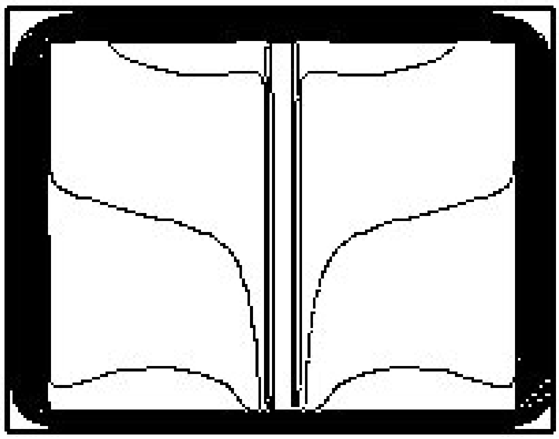


Figure 4 - Temperature distribution within the cavity.

An additional potential of the model was examined: the south surface was made adiabatic and Figure 5 shows the isotherms for such case. The fact that the isotherms are perpendicular to the south wall points out an adiabatic surface. Provided the flow is the same of Figure 2, the temperature distribution is not a representative one; the airflow is not symmetric because of boundary conditions asymmetry.

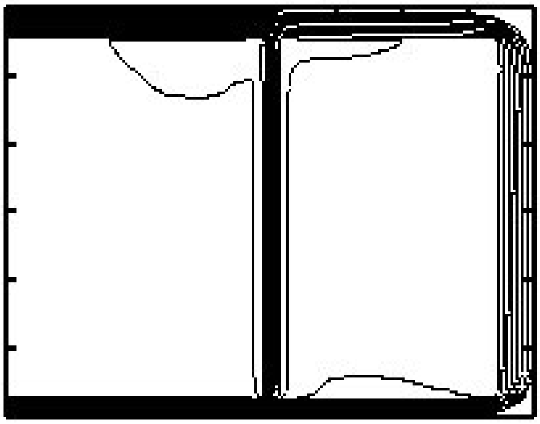


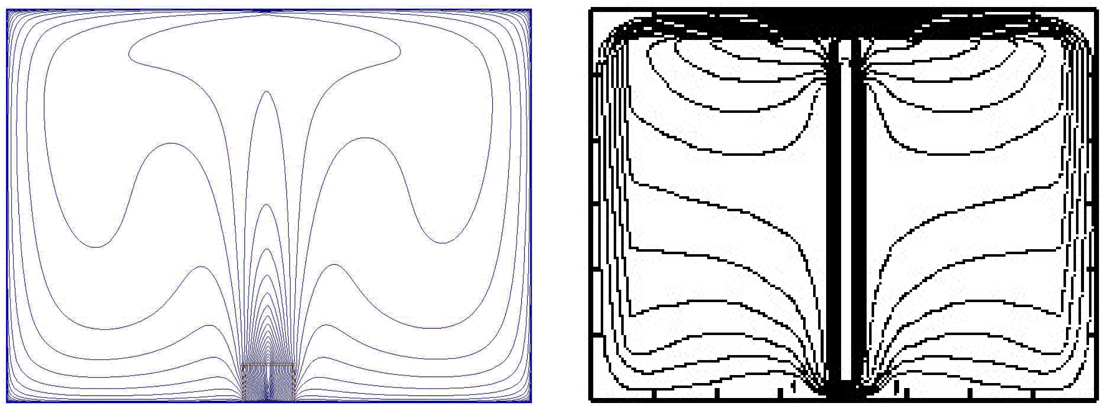
Figure 5 – Air temperature distribution for the adiabatic south wall.

In order to corroborate the results, an inter-model comparison was conducted. The CFD modelling was considered and the Flotherm™ (1998) package was employed as the base model. The Flotherm™ model was built for the geometry of Figure 1.

The Flotherm™ (1998) solves the steady 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. Flotherm™ model claims to provide a better estimate of gradients near the walls. The equations are discretized by the SIMPLE method by Patankar (1980). The employed Cartesian mesh of 164x112 points (18,368 control volumes) is refined where the higher gradients are expected; namely, close to the source and walls. A SUN Ultra Enterprise 450 Workstation (2 SUNW processors, 296MHz, 768Mbytes RAM) was employed in the CFD simulation. The necessary CPU time was about 10 min.

Figure 6a shows the isotherms obtained by Flotherm™. As expected, the highest temperatures take place at the heat source and the highest temperature gradients are close to the walls. Figure 7 illustrates the Flotherm™ velocity field which states a clear flow above the source and close to the walls. As foreseen, the velocities in the central region of the room assume low values and definitely only diffusion takes place in that place.

With the purpose to compare the models isotherms, some information necessary for VEM was obtained directly from the Flotherm™ results. The plume velocity was acquired from the Flotherm™ results; Figure 7 shows it is in the order of 0,3m/s. The width/height of the first row of cells close to the wall was also estimated from Figure 8 (width of cells near the vertical walls = 0,6m; height of cells near the top wall = 0,45m and near the bottom wall = 0,3m). The convection heat flow at the walls (Qs=Qn=205,2W, Ql=243W, Qh=296,4W) was computed based on the Flotherm™ results and they were imposed at the walls.



(a) (b)

Figure 6 - Comparison of isotherms produced by (a) Flotherm™ and (b) the current model.

Figure 6 shows that the isotherms of Flotherm™ are quite similar to those of the current model. The range of temperature within the cavity varies from 25 to 34.7°C in the VEM results and from 25.5 to 32.4°C in the CFD results. Both present a plume above the heat source. Also, the highest temperature gradients are near the walls and close to the heat source (the isotherms are closest to each other) in both cases. 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 (Figure 7) 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.

DISCUSSION AND CONCLUSIONS

The current work presents a 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.

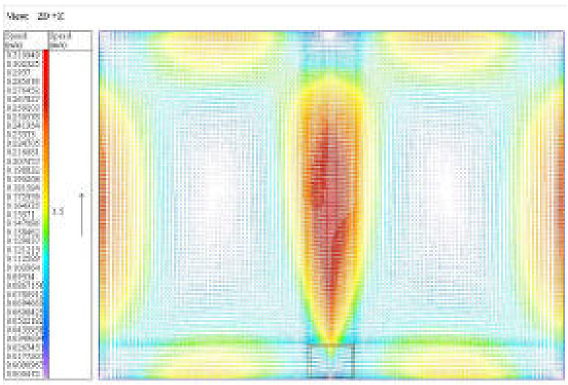


Figure 7 – The velocity field obtained with the k-ε revised turbulence model of Flotherm™.

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 equation (4) is also necessary to be conducted. If the Flotherm results can be considered accurate a 0.43 constant value could multiply equation (4).

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®, 1998, *Online Documentation.*

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*, pp. 155-165, Vol 27-2.

Patankar, S. V., 1980*, Numerical Heat Transfer and Fluid Flow*, Taylor and Francis.

Rodi, W. 1984, *Turbulence Models and Their Applications in Hydraulics – A State of The Art Review*, University of Karlsruhe, Karlsruhe, Germany

Vargas, J. V. C., 2001, Vargas, J. V. C., 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*

Table 1 – Temperature within the cavity in ^{o}C

x(m) / y(m)	0.30	0.80	1.20	1.60	1.85	1.95	2.05	2.15	2.40	2.80	3.20	3.70
2.775	33.76	34.07	34.19	34.31	34.43	35.37	35.37	34.43	34.31	34.19	34.07	33.76
2.425	33.69	33.85	33.91	33.86	33.55	35.44	35.44	33.55	33.86	33.91	33.85	33.69
2.175	33.62	33.73	33.76	33.70	33.55	35.44	35.44	33.55	33.70	33.76	33.73	33.62
1.925	33.55	33.62	33.65	33.62	33.55	35.44	35.44	33.55	33.62	33.65	33.62	33.55
1.675	33.48	33.53	33.56	33.56	33.55	35.44	35.44	33.55	33.56	33.56	33.53	33.48
1.425	33.41	33.44	33.48	33.52	33.55	35.44	35.44	33.55	33.52	33.48	33.44	33.41
1.175	33.34	33.35	33.39	33.47	33.55	35.44	35.44	33.55	33.47	33.39	33.35	33.34
0.925	33.27	33.25	33.29	33.41	33.55	35.44	35.44	33.55	33.41	33.29	33.25	33.27
0.675	33.20	33.14	33.15	33.30	33.55	35.44	35.44	33.55	33.30	33.15	33.14	33.20
0.425	33.14	32.99	32.97	33.09	33.55	35.44	35.44	33.55	33.09	32.97	32.99	33.14
0.150	32.90	32.80	32.70	32.60	33.55	35.44	35.44	33.55	32.60	32.70	32.80	32.90