Introduction of Specific Laws in Zonal Model to Describe Temperature Fields and Air Flow Patterns in Mixed Ventilated Buildings

Etienne Wurtz, Marjorie Musy and Laurent Mora

Laboratoire d'Etude des Phénomènes de Transfert Appliqués au Bâtiment,
Université de La Rochelle, France

(received on 12 Aug.1999; accepted on 5 Oct.1999)

Abstract

In this paper, a zonal model used to predict the air movement, temperature distribution and air quality in a room is presented. It is based on a rough partitioning of the room: It is an intermediate approach between one-node models (that consider an homogeneous temperature in each room, and, for that reason, do not permit to predict the thermal comfort in a room) and CFD models (that require great amount of simulation time). Where plumes, jets or thermal layers occur, air flow is described by empirical laws. In low velocity domains, flow rate are calculated in respect to the pressure distribution. This air flow model is coupled with a building envelope model including the calculation of radiant and conductive exchanges. The complete model is implemented in an object oriented environment, SPARK, in which modelling a room consists in connecting the different pre-design elementary models. Hence, the way of modelling is very modular, so that the zonal model can now be applied to a very large field of configurations. The strict syntax of SPARK permit having the simulation automatically generated. Now, imagining the creation of a very flexible tool that allows to represent a whole building is realistic.

Results of simulations in a 3D-room will be given and it will be shown that this model yields rather accurate results even with a rough partitioning.

KeyWords: zonal method, object oriented environment, air quality, simulation, plume, jet

Introduction

Room convection models are needed that are intermediate between single-air-node models, which give no information about air flow patterns, and CFD models, which give detailed temperature and flow distributions but are extremely compute intensive. Such intermediate models execute much faster than CFD calculations but give more accurate heat transfers than the single-node approach and provide temperature and flow distributions that are accurate enough to predict thermal comfort.

One such intermediate approach is the so-called "zonal model." In zonal models the inside of a room is divided into a small number of zones or "cells," which are usually rectangular parallelepipeds. (The number of cells is usually on the order of 10 to 100, com-

pared to 1000 or higher for typical CFD calculations.) Mass balance and heat balance equations are applied to the cells and the exchanges are calculated between them. The solution of the resulting set of coupled equations gives the air flow and temperature distribution in the room.

The first zonal models ([Leb78], [Lar80], [How85], [Ina88]) were based on fixed air flow directions and on the application of specific flow laws for plumes, jets and boundary layers. [AI92] have reviewed these models. Assuming fixed air flow directions obviously restricts the field of application of these models.

Other models ([Bou93], [WNW96]) made the inter-cell air flow rates a function of the pressure distribution. It has been shown that this approach cannot correctly represent

the driving flows [MWG96].

Later, hybrid models were formulated that applied specific laws for driving flows and used a power-law pressure distribution everywhere else ([Gre87], [BD91]). However, these models are applicable only to a few simple configurations.

In the present work we present an advanced formulation of a zonal model and show how the resulting set of equations can be solved with the Simulation Problem Analysis and Research Kernel (SPARK), an object-oriented simulation environment [BEN+93]. To achieve a complete thermal calculation we have connected the zonal air flow calculation with conduction through the room walls and radiant interchange between them. This approach allows virtually any geometrical configuration to be modeled.

1. The Zonal Method

1.1 Previous Work

The zonal method, which is not new, is based on heat and mass balance equations in macroscopic volumes. Added to this is a relationship between mass flow and pressure difference. The zonal method should not be confused with models used to calculate air flow between rooms ([Wal84], [Oku87], [FA89], [HC91], [TS93]).

Initial work on the zonal method emphasized how to partition the computational domain in two dimensions ([LN87], [How85], [Ina88]). A systematic attempt to use the zonal method with power law equations on arbitrary geometry in two dimensions is described in [BD91], in which convergence problems were encountered and the results did not agree well with measurements.

The present work treats the 3-D simulation case, with natural or mixed convection. Results were validated against CFD calculations and measurements. In addition we coupled 3-D zonal models with models for thermal comfort, wall conduction and directed

flow. Other studies ([RAC93], [RAC94]) have considered the 3-D case, but did not couple to other models and were not validated.

1.2 Presentation of the method

In the 3-D zonal method the room air is partitioned into 3-D zones. Adjacent zones exchange mass and energy. The following mass and energy balance equations apply to each zone:

$$\begin{aligned}\sum q_m + q_{source} - q_{sink} &= 0 \\ \sum \Phi + \Phi_{source} - \Phi_{sink} &= 0\end{aligned}$$

where $\sum q_m$ is the sum of mass flows across the interfaces of the zones, q_{source} is the mass flow from sources in the zone, q_{sink} is the mass flow to sinks in the zone, $\sum \Phi$ is the sum of heat transfers through the interfaces of the zone, Φ_{source} is the heat transfer from sources in the zone and Φ_{sink} is the heat transfer to sinks in the zone.

We assume that a zone has uniform temperature and density and that the pressure at the middle of a zone obeys the perfect gas law:

$$P_{middle} = \rho r T$$

where ρ is the density, r is the air molar constant and T the absolute temperature.

The pressure at height z above the bottom of a zone is given by:

$$P = P_0 - \rho g z$$

where P_0 is the pressure at the bottom of the zone.

Assuming that the zones are rectangular parallelepipeds with edges oriented along the coordinate axes, the gist of the zonal model lies in the following equations:

- The elementary mass flow rate dq_m flowing to the elementary flow is expressed as follows:

$$dq_m = C \rho \Delta P^n dS$$

- For vertical interfaces, depending on the position of the neutral axis z_n and the

value of p , we have to break down the flow rate q_m into two flows rate $q_{m\sup}$ and $q_{m\inf}$.

$$\begin{aligned}\Delta P &= \Delta P_0 - \Delta \rho g z \\ z_n &= \frac{\Delta P_0}{\Delta \rho g} \\ q_{m\sup} &= C l \rho (\Delta \rho g)^n \frac{(h - z_n)^{n+1}}{n+1} \\ q_{m\inf} &= C l \rho (\Delta \rho g)^n \frac{(z_n)^{n+1}}{n+1}\end{aligned}$$

The flow exchanged becomes:

$$q_m = q_{m\sup} + q_{m\inf}$$

- For horizontal interfaces, the pressure differential is uniform at the border level and there is only one flow rate:

$$q_{m_{\text{vert}}} = C \rho S (P - P_{\text{top}})$$

Here P_{top} is the pressure at the top of the zone, q_m are mass flows, C is an empirical constant equal to $0.83 \text{ ms}^{-1} \text{ Pa}^{-n}$ [Ina88] (A parametric study [Wur95] has shown the validity of this parameter), l is the width of a zone, h is the height of a zone, n is a fractional exponent (0.5 for turbulent air flow and 1 for laminar air flow), z_n is the height of the neutral point (the point at which pressures on either side of the interface between two zones are equal), ρ is the volumetric mass, vert is an index for vertical mass flow, $m_{\sup}$ an index for horizontal mass flow above the neutral point and $m_{\inf}$ is an index for horizontal mass flow below the neutral point. It is assumed that the temperature in each zone is uniform, flow is incompressible and pressure drop is hydrostatic.

Energy fluxes are calculated using the following equations, which are valid for temperatures typically found in buildings and which, for now, neglect humidity:

$$\begin{aligned}\Phi_{\text{horiz}} &= q_{m_s} C_p T_s + q_{m_e} C_p T_e \\ \Phi_{\text{vert}} &= q_{m_{\text{vert}}} C_p T_{\text{vert}}\end{aligned}$$

where Φ denotes a heat flux. Here

q_{m_s} denotes mass flow leaving the zone and q_{m_e} denotes mass flow entering the zone. T_s is the temperature of the air leaving the zone, which is the same as the zone temperature, and T_e the temperature of the air entering the zone. T_{vert} is the zone temperature for vertical flow out of the zone and is the adjacent zone temperature for vertical flow into the zone. We use a sign convention such that a mass flow is positive when the flow is in the positive direction of an x , y or z axis, and negative when in the negative direction of an axis.

If the standard area represents the surface of a partition, a convective exchange takes place which can be modeled by a coefficient h_{conv} . The heat flow then reads:

$$\Phi_{\text{conv}} = h_{\text{conv}} S (T - T_w)$$

T_w is the surface temperature of the wall.

1.3 Physical Considerations

Several remarks can be made about the validity of the model. (1) Coarse grids and high temperature gradients may make uniform temperature in a zone a poor assumption. (2) Temperature and velocity boundary layers are not accounted for. (3) The hydrostatic pressure approximation is valid only for flows with parallel streamlines. (4) Only one neutral point per vertical interface between zones is allowed, which, depending on the grading, affects the qualitative behavior of the solution. (5) Along the lines of [BD91], the Bernoulli equation used amounts to assuming that kinetic energy is fully dissipated within the bounds of a single zone, and so does not apply to plumes or jets that span two or more zones, as rediscovered in [RAC93].

1.4 Numerical Considerations

An obvious property of the above equations is their nonlinearity. The $n = 0.5$ exponent is a source of particular numerical difficulty since it is well known that standard Newton-Raphson iteration (without relax-

ation) does not converge when there are square-root dependencies. An additional source of trouble is that, because they depend on flow direction, the equations must be piecewise defined, even if they are formally unified, as in [HvdMR93]. Finally, 3-D problems generally lead to a fairly large number of equations. This “dimensional curse” also exists for zonal models. For example, some problems analyzed with zonal methods require resolving more than 2000 equations.

2. Numerical Implementation: the SPARK Environment

2.1 Presentation and History

The Simulation Problem Analysis and Research Kernel (SPARK) is a modular environment that automates writing code for systems of non-linear equations. It was developed for building science but is applicable to other fields. It is related to simulation environments like TK!Solver [KJ85], TRNSYS [Sol88], CLIM2000 [BCGR89], IDA [Sah88], and Allan.Simulation [Fra92].

Some key features of SPARK are:

- It has a front end that allows the user to build complex simulations by connecting smaller elements that are objects (single equations) or macro-objects (equation subsystems). It shares this feature with TRNSYS, CLIM2000 and Allan.Simulation.
- Using graph-theoretic techniques, it reduces the size of the equation system by automatically determining a small set of iteration variables for which the other unknowns can be solved. This step can be viewed as “smart” elimination of variables. SPARK's Newton-Raphson solver works on the reduced equation set and, after convergence, the remaining unknowns are solved for. This is a unique feature. Allan.Simulation, for example, generates code that inverts the full Jacobian matrix.
- Its output is a C program that is automatically compiled and executed. This pro-

gram accepts user-specified input at run time and is calculatingly efficient because it iterates on a reduced set of variables.

- Passing from a simulation problem to a design problem (i.e., having unknowns become inputs and inputs become unknowns) is simply a matter of keyword exchange in SPARK.

Originally written in 1986 for steady state problems [And87], it was extended in 1989 to handle transient problems by adding time integrator objects [SBN89]. Recent enhancements include

- Automatic generation of objects from equations expressed symbolically [SNW90], [NW94]).
- Strong component decomposition to reduce execution time [BEN+93].
- User control of solution method to enhance convergence.

2.2 Implementation of the Zonal Model

Implementation of zonal models in SPARK is straightforward. The main object classes correspond to the *zones* and to the *interfaces* between *zones*. A *zone* class consists of the balance equations for the zone, the pressure drop equation and the perfect gas law. An *interface* class consists of the neutral point calculation and the relationship between mass flow and pressure difference.

These classes are instantiated as many times as needed to define the simulation. For example, if a 3-D room is divided into eight parallelepipeds (two in each of the x , y and z directions), there will be eight zone objects and 36 interface objects (12 zone-to-zone interfaces and 24 zone-to-surface interfaces). In the general 3-D case, if the x , y and z axes are divided into L , M and N sections, respectively, there will be $L M N$ zone objects and $3(L M + M N + L N)$ interface objects. In 2-D there will be $L M$ zone objects and $L M$ zone-to-zone interface objects and $2 L M$ zone-to-surface interface objects.

After instantiation, the objects are linked, i.e., the variables shared by objects are

identified. Then the objects and their linked variables are stored in a file that specifies the overall problem and its inputs.

2.3 Efficiency

The efficiency of a simulation environment depends on the time and resources needed to solve a simulation problem. SPARK obtains a near-optimal simplification of the problem, without loss of precision, by automatically reducing the number of iteration variables. The reduction can be more than 10-fold, which corresponds to a roughly 1000-fold decrease in Jacobian inversion time. SPARK also makes it easy to create a simulation from scratch by using symbolic processing to create the simulation building blocks (equation objects) from equations written by the user [NW94]. In the present work even the linking of objects was automated [NW93],[Wur95]).

To facilitate interpretation of results, a graphical postprocessor was written to visualize temperatures and air flows.

3. Integration of specific models

Wurtz showed [Wur95] that zonal models were only able to describe air flows where the velocity is low, and demonstrated the possibility to implement particular air flows models in the object-oriented simulation environment SPARK. Finally the incorporation of specific air flows models, as plume, boundary layer and jet models in simulations was done by Musy [Mus99], who showed their efficiency.

Fig. 1: Cells associated with the plume model

3.1 Plume Model

3.1.1 Modeling

A plume is a region of rising warm air such as that produced by a convective heater next to a wall (figure 1). We assume that a plume does not develop laterally as it rises but entrains air through its side faces.

The plume model incorporates five calculation objects. The heater is represented by a *heater cell*. A *heater/wall interface* describes the thermal exchange between the back of the heater and the wall. The plume itself is represented by a *plume cell*, a *horizontal plume interface* and a *vertical plume interface*:

Fig. 2: A plume cell and its interfaces. A plume cell contains two subcells and their corresponding subinterfaces. One of these subcells (shaded) contains air from the plume and the other (unshaded) contains surrounding air.

A plume cell contains two subcells, one containing air belonging to the plume itself and one containing air from the surroundings. This type of partitioning is also done for a heater cell. For this reason, the plume and heater objects consist of equations that are reused from the basic cell object (to represent the surrounding air) plus additional equations that are specific to the plume or heater. These objects link to their neighbors in the same way that basic cells link except that there are additional links.

Note that the heater and its associated plume are contained in rectangular parallelepipeds of the same size and shape stacked on top of each other. This simplifies partitioning of a room that has plumes since it makes

it easy to “line up” the plume and heater cells with the other cells in the room.

In the following we describe the thermal characteristics of a plume using the equations of Inard *et Al.* [IMD+1997]. For horizontal interfaces the plume width is

$$Bu = \frac{2E_0}{\sqrt{\pi}} \left(1 + \frac{St(1+\lambda^2)^{1/2}}{3\lambda E_0} \right) (z - z_0)$$

The maximum plume velocity is

$$Um = \left(\frac{6\lambda(1+\lambda^2)^{1/2}}{3\lambda(Cf + \sqrt{2}E_0) - (2(1+\lambda^2))^{1/2}St} \right)^{1/3} \left(\frac{\phi g \beta}{\rho_0 C p_0} \right)^{1/3}$$

where $\phi(z)$ is the horizontal heat flux into a plume cell.

The plume air flow rate is

$$q = \frac{\sqrt{\pi}}{2} \rho_0 Um Bu$$

The maximum temperature difference between the midpoint of a plume cell and the mean room air temperature is given by

$$\Delta Tm = \left(\frac{\phi g \beta}{\rho_0 C p_0} \right)^{2/3} \left(\frac{3}{2St(1+\lambda^2)^{1/2} + 6E_0} \right) \left(\frac{1}{g\beta(z - z_0)} \right) \left(\frac{4(1+\lambda^2)}{3\lambda} \left(3\lambda(Cf + \sqrt{2}E_0) - (2(1+\lambda^2))^{1/2}St \right) \right)^{1/3}$$

Here $\phi(z)$ is the average of the heat fluxes crossing the upper and lower interfaces of the subcell that contains the plume (see figure 2). These heat fluxes are determined using the energy balance equation applied to the subcell. Similarly, the air flow rate is determined by applying the mass balance equation to the subcell. The mass balance equation also allows the flow rate of air entrained by the plume to be calculated. This flow rate is assumed to be the same on all faces (front and lateral) of the plume.

The top of the plume (i.e., the point at which the plume has dissipated) is defined to be the height at which $\phi(z)$ becomes less than a threshold value. Above this height, Bu, Um and ΔTm are set to zero and the cells revert back to basic cells. Therefore, it is not

necessary to know a priori how high the plume will reach: the simulation will automatically convert plume cells to basic cells above the dissipation point.

For the heater cell, the temperatures of the front and back surfaces of the heater (figure 3) are given by the following equations [Bar92], [Bar93]:

$$T_{front_face} = 0.67 \left(\frac{P_{elec}}{P_{nom}} \Delta T_{s_{max}} \right) + Ta$$

$$T_{back_face} = \frac{P_{elec}}{P_{nom}} (0.67 \Delta T_{s_{max}} - 7) + Ta$$

Fig. 3 : Heater model

The sum of the convective and radiative flux from the heater is set equal to the heater input power:

$$P_{elec} = \Phi_{conv_back_face} + \Phi_{conv_front_face} + \Phi_{rad_back_face} + \Phi_{rad_front_face}$$

It is assumed that the convective flux from the heater goes entirely into the plume.

The back face of the heater is assumed to radiate only to the wall behind the heater, giving

$$\begin{aligned} \Phi_{net_wall} &= -\Phi_{net_back} \\ &= \frac{\epsilon_{heater} \epsilon_{LW_wall} S \sigma (T_s^4 - T_{back_face}^4)}{\epsilon_{heater} + \epsilon_{LW_wall} - \epsilon_{heater} \epsilon_{LW_wall}} \end{aligned}$$

The front face of the heater radiates to the other surfaces of the room.

The convective exchange along the wall behind the heater is calculated using the

Fig. 5 : Calculated temperature and air flow values in the East-West vertical plane passing through the heater (see figure 4). The left-hand plot shows the results of the calculation done without modeling the plume produced by the heater. On the right side of this plot there is a heater cell at the bottom with three basic cells above it. In the right-hand plot the plume has been explicitly modeled. On the right side of this plot there is a heater cell at the bottom with two plume cells above it, followed by a basic cell at the ceiling. On the left side of both plots there are four basic cells from floor to ceiling. Shading shows the different temperature levels. Air flow rate is proportional to arrow thickness.

Fig. 6 : Comparison of the measured temperature along the vertical centerline of the CETIAT test room with the results of the simulation when using the plume model. The calculated temperatures are assumed to be uniform in each cell and are therefore shown as horizontal line segments. In this plot the centerline temperatures (solid triangles) have been calculated as the mean of the Section 1 and Section 2 temperatures (see right-hand plot in figure 5). The air temperature increases by about 7°C from floor to ceiling, indicating that there is significant stratification. For comparison, the temperature difference between the ceiling surface and floor surface is only $23.6 - 21.1 = 2.5^{\circ}\text{C}$

Fig. 7 : Boundary layer model : simulation results and comparison with experimental data

3.2 Boundary Layer

3.2.1 Modeling

The boundary layer model applied is proposed by Allard *et al.* [AIS90], and express air flow rate as a function of temperature difference between wall surface and air:

$$q_m(z) = 0.004(T_w - T)^{1/3} \cdot z$$

where z is the distance between the considered point and boundary layer origin point. T is air temperature near the wall, and T_w is wall surface temperature where is growing up a boundary layer.

Generally, between heights z and $z+dz$, we can express air flow rate as:

- $q_m(z + dz) = q_m(z) + 0.004(T_w - T)^{1/3} dz$

if boundary layer air flow going up words

- $q_m(z) = q_m(z + dz) + 0.004(T_w - T)^{1/3} dz$

if boundary layer air flow going down words

we can reduce that to:

$$q_m(z + dz) = q_m(z) + 0.004 \left| T_w - T \right|^{1/3} dz$$

3.2.2 Results

The simulation results were compared with measurements from CSTB EREDIS experimental cell. It's a squared room, 3.6 m length and 2.5 m height. All surfaces have same homogeneous temperature excepted one that is a glazing with 3 different temperature points. Figure 7 compares simulation results and measurements.

3.3 Jets

3.3.1 Modeling

A jet cell contains two subcells, one containing air belonging to the jet itself and one containing air from the surroundings. For this reason, the jet objects consist of equation that are reused from basic cells objects (to represent the surrounding air) plus equations that are specific to the jet. Note that the jet is

contained in rectangular parallelepipeds set side by side (figure 1). This simplifies the partitioning of rooms that have jets since it make it easy to line up the jet cells with the other cells.

In the following, we describe the characteristics of a two-dimensional isothermal ceiling jet using the equation of Schwarz *et al.* [SC61] and Rajaratnam [Raj76]. The jet width, $h(x)$, the maximum jet velocity $U_{mj}(x)$, and the air flow rate, $Q_{mj}(x)$, are:

$$h(x) = 2.36 C_j x$$

$$U_{mj}(x) = U_{mj0} K_v \sqrt{\frac{h_0}{x}}$$

$$Q_{mj}(x) = 1.054 C_j K_v Q_{mj0} \sqrt{\frac{x}{h_0}}$$

U_{mj0} and Q_{mj0} are the velocity and the air flow rate in the diffuser, h_0 the diffuser height. K_v and C_j are empirical coefficients, they are respectively set to 3.5 and 0.068

These equations are the additional equations of vertical jet interfaces.

The mass balance equation, applied to the jet part of the cell allows the flow rate of air entrained by the jet to be calculated. The end of the jet is the interface in which $U_{mj}(x)$ becomes less than a threshold value. In this interface and in the following ones $h(x)$ and $Q_{mj}(x)$ are set to zero and the cells revert back to basic cells. Therefore, it is not necessary to know *a priori* how long the jet will reach: the simulation will automatically convert jet objects.

Now, if we consider that the air consists of several gaseous species, mass balance equation must be written for each of them:

$$Q_{mWest_j} - Q_{mEast_j} + Q_{mSouth_j}$$

$$- Q_{mNorth_j} - Q_{mTop_j} + Q_{mBottom_j}$$

$$= Q_{mBottom_j} - \rho h L w \frac{dX_j}{dt}$$

X_j is the ratio of the specie density and the air density which is equal to the sum of the species' densities. The source term, $Q_{msource_j}$ can be an input data or calculated by using laws describing the chemical kinetic of the reactions between the different species.

The movement of the species in the room is governed by the two phenomena that are transport and diffusion. Mass flow rates are the sum of two terms corresponding to both of them. Therefore, the mass flow rate of specie 'j' across the east face, for example is given by:

$$Q_{mEast_j} = -Lh \left(CX_{i_j} (P_i - P_{East}) + 2D_j \frac{X_{i_j} - X_{East_j}}{l_{East} + l_i} \right)$$

D_j is the specie's diffusion coefficient (for the solvent its value is zero).

Perfect gas law is kept as well as equations about heat exchanges.

3.3.2 Results

The simulation results will be compared with measurements from the CETIAT experimental room, MINIBAT [Cas98], equipped with a ventilation device. Figure 9 shows the room geometry and partitioning into cells, and the location of the diffuser. The inputs to the simulation are the inside surface temperatures, the entering air flow rate, and a pollutant source (SF6) in the middle of the room (Table 2).

The simulation results were compared with measurements from the CETIAT experimental room. Temperatures agree with measurement to within 1°C in the three bottom layers and to within 1.3°C in the top layer. Pollutant concentration is too low excepted near the source. This could be attributed to the fact that for the moment, the model doesn't take into account a turbulent diffusion coefficient that can in some place of the room be high compared to molecular diffusion coefficient.

Fig. 8: Cells associated with the jet model

Fig. 9: The MINIBAT experimental room partitioned into $4 \times 3 \times 4 = 48$ cells. Three of these are jet cells.

Table 2. Input values for inside surface temperature, inlet air and pollutant source.

	Floor	Ceiling	East	South	North	West	Inlet	Pollutant
Temperature(°C)	19.4	21.	20.	18.9	20.	19.9	33.5	
Air flow rate (kg/s)							8.32×10^{-3}	2.943×10^{-3}

Fig. 10 : Calculated temperature in the South-North vertical plane passing through the jet. Shading shows the different temperature or pollutant density levels. Air flow rate is proportional to arrow thickness.

Table 3: Values of the coefficient and exponent in the expression for convective heat transfer coefficient, for h_{conv} in $W.m^{-2}.K^{-1}$ and temperature difference in K.

		C_{conv}	n_{conv}
Ceiling	natural convection [Ala87]	3.0	0
	mixed convection [Ina88]	3.0	0.7
Roof	heating roof [KK82]	1.5	1/3
	other cases [Ala87]	3.0	0
walls	natural convection [Ala87]	1.5	1/3
	mixed convection [Leb70]	3	1/3
	plume aft face [Ina88]	0.662	0.77

Fig. 11: Fictitious-surface method for calculating radiant exchange among room surfaces. A wall with temperature T_{si} exchanges radiation with a single fictitious surface with temperature TF_i .

4. Models coupling

4.1 Wall model

Wall conduction, assumed to be one dimensional, is calculated with finite difference equations. The convective flux at the inside wall surface is

$$\phi_{conv} = h_{conv} (T_i - T_{wi})$$

where the convective heat transfer coefficient, h , depends on the surface-to-air temperature difference according to:

$$h_{conv} = C_{conv} (T_i - T_{wi})^{n_{conv}}$$

for which the values of the coefficient, C_{conv} , and exponent, n_{conv} , are given in Table 3.

4.2 Radiant exchange model

We use Walton's method [Wal80] to calculate long- and short-wave radiant exchange among the inside room surfaces. In this method each room surface is assumed to radiate to a fictitious surface (figure 11) whose area, emissivity and temperature give about

the same heat transfer from the room surface as in the actual multi-surface case. The advantage of the method is that it considerably reduces the number of interchange equations.

For long-wave radiation the area of the fictitious surface associated with room surface i is the sum of the areas of all surfaces except surface i :

$$SF_i = \sum_{j=1}^n (1 - \delta_{ij}) S_j$$

The emissivity of the fictitious surface is the area-weighted average of the emissivities of the other surfaces:

$$\epsilon F_i = \frac{\sum_{j=1}^n (1 - \delta_{ij}) \epsilon_j S_j}{SF_i}$$

The temperature of the fictitious surface is the area-times-emissivity weighted-average temperature of the other surfaces:

$$TF_i = \left(\frac{\sum_{j=1}^n (1 - \delta_{ij}) \epsilon_j S_j T_{s_j}}{\epsilon F_i SF_i} \right)$$

Finally, the net radiant energy per unit area absorbed by surface i is given by:

$$\phi_{i,net} = \sigma K_i (T_{s_i}^4 - TF_i^4)$$

where

$$K_i = \frac{1}{\frac{1}{\epsilon_i} + \frac{S_i}{SF_i} * \frac{1 - \epsilon F_i}{\epsilon F_i}}$$

4.3 Dynamic multizone simulations

A dynamic term is added to thermal balance equation, that becomes:

$$\begin{aligned} \Phi_{East} - \Phi_{West} + \Phi_{North} - \Phi_{South} + \Phi_{Top} - \Phi_{Bottom} \\ = \Phi_{source} - h.w.L.C_p.\rho_i.\frac{\partial T_i}{\partial t} \end{aligned}$$

Fig. 12 : Calculated temperature and air flow values in the East-West vertical plane passing through the heaters. Shading shows the different temperature levels. Air flow rate is proportional to arrow thickness. Note that this is a section of the room, it is why the mass balance equation seems not to be respected in some cells.

4.4 Results

The simulation is a four rooms building. Rooms 1 to 3 are identical. Room 4 is an atrium. Their dimensions are given in figure 13. In South-North direction the thickness of the building is 3 meters. Opened doors separate each small room from the atrium. Rooms 1 to 3 contain heaters. The model constructed consists in 108 cells ($2 \times 3 \times 3$ in rooms 1, 2, 3 and $2 \times 3 \times 9$ in room 4). In each small room, there is an heater cell and a plume cell. The other cells are basic ones. The input data are the wall external surface temperature, the outside air temperature, the power of the heaters and inlet flow rates.

The results in the East-West vertical plane passing through the heaters are given in figure 12.

One can note that in the atrium there is stratification (about 4°C) and that mass exchanges through the doors are significant (mass flow rates are about 6 times greater than the inlet flow rates). The results seem qualitatively correct.

Conclusion

We have shown how a zonal model can be used to calculate room air temperature and air flow distributions. In the cases studied, dividing the room air volume into 24 or 45 cells was sufficiently fine-grained to allow thermal comfort to be determined without excessive calculation time or machine capacity. This will encourage us to use the new PC version of SPARK.

Different levels of automation were used to generate models in the SPARK object-oriented simulation environment:

- Generating equation objects using symbolic processing.
- Automatic generation of SPARK simulation files for a whole building.

This make it possible to generate a complete simulation of a building given only

the building geometry and properties of the heating and cooling systems.

We are currently working on integrating other models for particular flows to allow simulating air conditioning systems. We are also working on taking into account pollutants in the air flow model.

So far, the comparisons we have made with experimental measurement have been encouraging. We are currently making additional comparisons to further validate our zonal model approach.

It is important to note that the simulation approach that we have developed can be applied to all types of thermal and air-flow problems: it is not restricted to simple configurations nor is it required that the flow distribution be known *a priori*.

References

- [Ala87] F. Allard. *Contribution à l'étude des transferts de chaleur dans les cavités thermiquement entraînées : application aux cellules d'habitation (Contribution to the study of heat transfer in thermally entrained cavities: application to rooms)*. Thèse de Doctorat ès Sciences, INSA de Lyon, France. 1987.
- [AI92] F. Allard and C. Inard. *Natural and mixed convection in rooms: prediction of thermal stratification and heat transfer by zonal models*. International Symposium of Room Air Convection and Ventilation Effectiveness. University of Tokyo, July 22-24, 1992.
- AIS90] F. Allard, C. Inard and J.-P. Simoneau. *Phénomènes convectifs intérieurs dans les cellules d'habitation. Approche expérimentale et numérique. Revue Générale de thermique*, 29 (340), pp 216-225, 1990.

- [And87] J. L. Anderson. A network definition and solution of simulation problems. Technical Report 21522. Lawrence Berkeley National Laboratory, Berkeley, CA 94720. September 1987.
- [Bar92] P. Barles. *Carcatérisation de l'émission des convecteurs électriques - Etude statistique (Characterization of the output of electric convective heaters)*. Centre Technique des Industries Aérauliques et Thermiques, CETIAT/ADEME contract report (2.04.005), 1992.
- [Bar93] P. Barles. *Contribution du CETIAT aux travaux du sous-groupe convecteurs électriques du GREC (Contribution of the CETIAT to the work of the GREC subgroup on electric convective heaters)*. Centre Technique des Industries Aérauliques et Thermiques, ADEME contract final report (2.04.005), 1993.
- [BCGR89] D. Bonneau, D. Covalet, D. Gautier, and F. X. Rongere. Manuel de prise en main de CLIM2000, version 0.0, 1989.
- [BD91] H. Bouia and P. Dalacieux. Simplified modeling of air movements inside dwelling room. In *Proceedings of Building Simulation '91*, Sophia-Antipolis, Nice, France. International Building Performance Simulation Association, 20-22 August 1991.
- [BEN+93] F. Buhl, E. Erdem, J.M. Nataf, F. Winkelmann and E. Sowell. *Recent improvements in SPARK: Strong component decomposition, multivalued objects and graphical interface*. Lawrence Berkeley National Laboratory Report LBL-33906. August 1993.
- [Bou93] H. Bouia. *Modélisation simplifiée d'écoulements de convection mixte internes: application aux échanges thermo-aérauliques dans les locaux*. Ph. D. Thesis, University of Poitiers, France, 1993.
- [Cas98] S. Castanet. *Contribution à l'étude de la ventilation et de la qualité de l'air intérieur des locaux*, Ph. D. Thesis, INSA de Lyon, France. 1998.
- [Dur94] H. During. *Consommation énergétique et confort thermique des locaux chauffés: approche par les modèles zonaux (Energy consumption and thermal comfort of heated rooms: zonal model approach)*, Ph.D. thesis, Insa de Lyon, 1994
- [FA89] Helmut E. Feustel and Al. The COMIS infiltration model. In *Proceedings of Building Simulation 89*. Vancouver, June 23-24 1989. International Building Performance Simulation Association.
- [Fra92] Gaz de France. Allan. Manuel de référence, Allan 2.4.1. Technical Report, Gaz de France, June 1992.
- [Gre87] A. Grelat. *Approche des phénomènes de circulation et de stratification de l'air dans les locaux chauffés par le programme de simulation thermique multizone BILGA (Approach used by the multizone thermal simulation program BILGA to calculate air circulation and stratification phenomena in heated rooms)*. CEBTP-EDF contract final report, February 1987.
- [HC91] J. L. M. Hensen and J. A. Clarke. A simulation approach to the evaluation of coupled heat and mass transfer in buildings. In *Proceedings of Building Simulation '91*, Sophia-Antipolis, Nice, France. International Building Performance Simulation Association, 20-22 August 1991.
- [How85] A. T. Howarth. The prediction of air temperature variations in naturally

- ventilated rooms with convective heating. *Building Service Engineering Research and Technology*, 6(4): 169-175, 1985.
- [HvdMR93] J. Hensen, J. Van der Maas, and A. Roos. Air and heat flow through large vertical openings. In *Proceedings of Building Simulation '93, Adelaide, Australia*, pages 479-485. International Building Performance Simulation Association. 16-18 August 1993.
- [Ina88] C. Inard. *Contribution à l'étude du couplage thermique entre un émetteur de chauffage et un local*. PhD thesis, Institut National des Sciences Appliquées de Lyon, July 1988.
- [IMD+1997] C. Inard, A. Meslem, P. Depecker, P. Barles. *Structure moyenne et analyse intégrale du panache thermique des convecteurs électriques (Average structure and complete analysis of thermal plumes from convective electric heaters)*. *Revue Générale de Thermique* 36: 495-509, 1997.
- [KJ85] M. Konopasek and S. Jarayaman. Constraint and declarative languages for engineering applications: the tk!solver contribution. *Proceedings of the IEEE*, 73(12), December 1985.
- [KK82] Kast and Klan. *Auslegung und prüfung von fußbodenheizungen (Interpretation and test of floor heating)*. VDI-Berichte 464: 39-49. 1982.
- [Lar80] L. Laret. *Contribution au développement de modèles mathématiques du comportement thermique transitoire de structures d'habitation (Contribution to the development of mathematical models of the transitory thermal behavior of residential structures)*. Ph.D. Thesis, University of Liège. 330p. 1980.
- [Leb70] J. Lebrun. *Exigences physiologiques et modalités physiques de la climatisation par source statique concentrée (Physiological requirements and physical modalities of air conditioning by a static concentrated source)* Ph.D. Thesis, University of Liège. 1970.
- [Leb78] J. Lebrun *Etudes expérimentales des régimes transitoires en chambres climatiques : ajustement des méthodes de calcul (Experimental studies of the transition regime in climatic chambers: adjustment to calculation methods)*. Journées Bilan et Perspectives en Génie Civil. 1978.
- [LN87] J. Lebrun and P. Ngendakumana. Air circulation induced by heating emitters and corresponding heat exchanges along the walls: test room results and modelling. In *Proceedings of Room Vent 1987*, Stockholm, 1987.
- [Mus99] M. Musy. Génération automatique de modèles zonaux pour l'étude du comportement thermo-aéraulique des bâtiments. Ph. D. Thesis, July 1999.
- [MWG97] M. Musy, E. Wurtz, S. Gagneau. Génération automatisée d'une simulation thermique et aéraulique dans le bâtiment (Automatic generation of a thermal and air-flow simulation in a building). Journée « Innovations et perspectives de recherche », Société Française des Thermiciens, November 26, 1997.
- [NW93] J.-M. Nataf and E. Wurtz. Application of the Spark environment to 3D air flow problems. In *Proceedings of Buildings Simulation '93, Adelaide, Australia*. International Building Performance Association, August 1993.
- [NW94] J.-M. Nataf and F. Winkelmann. Symbolic modeling in building

- energy simulation. LBNL Report LBL-35439. Lawrence Berkeley National Laboratory, April 1994.
- [Oku87] Hiroyasu Okuyama. *Theoretical Study on the Building Thermal Network Model* (in Japanese). PhD thesis. Waseda University, December 1987.
- [RAC93] E. A. Rodriguez, S. Alvarez, and I. Caceres. Prediction of indoor temperature and air flow patterns by means of simplified zonal models. In *Proceedings of ISES Solar World Congress, 1993, Budapest, Hungary*, volume 6, 1993.
- [RAC94] E. A. Rodriguez, S. Alvarez, and J. F. Coronel. Modelling stratification patterns in detailed building simulation codes. In *Proceedings of European Conference on Energy Performance and Indoor Climate in Buildings, 1994, Lyon, France, 24-26 November 1994*.
- [Raj76] N. Rajaratnam, *Turbulent jets*. Amsterdam: Elsevier Scientific Publishing Company. 1976
- [Sah88] P. Sahlin. IDA, a Modeling and simulation environment for Building Applications. Institute of Applied Mathematics. P.O. Box 26300, S-100 41 Stockholm, Sweden, 1998.
- [SBN89] E. F. Sowell, W. Buhl and J.-M. Nataf. Object-oriented programming, equation-based submodels, and system reduction in spank. In *Proceedings of Building Simulation '89*, pages 141-146, Vancouver, British Columbia, Canada, June 1989. International Building Performance Association, P.O. 282, Oleans, Ontario, K1C 1S7, Canada.
- [SC61] W.H. Schwartz and W.P. Cosart. *The two-dimensional turbulent wall-jet*. J Fluid Mech., Vol 10, pp. 481-495, 1961.
- [SNW90] E. F. Sowell, J.-M. Nataf and F. Wilkelmann. Radiant Transfer due to lighting: An example of symbolic model generation for the simulation problem analysis kernel. LBL Report LBL-28273, Lawrence Berkeley Laboratory, January 1990.
- [Sol88] Solar Energy Laboratory, University of Wisconsin-Madison. *TRNSYSa Transient Simulation Program*, 1988.
- [TS93] T. Tsuchiya and K. Sakano. Computer simulation of multiroom temperature and humidity variation under variable infiltration conditions. In *Proceedings of Building Simulation '93, Adelaide, Australia*. International Building Performance Simulation Association, 16-18 August 1993.
- [Wal80] G.N. Walton. *A new algorithm for radiant exchange in room loads calculations*, ASHRAE Transactions, vol 86, Part II, pp 190-208; 1980
- [Wal84] George N. Walton. A computer algorithm for predicting infiltration and interroom air flows. *ASHRAE Transactions*, 90, 1984.
- [WNW96] E. Wurtz, J.M. Nataf and F. Winkelmann. *Two- and Three-Dimensional Natural and Mixed Convection Simulation Using Modular Zonal Models*. LBL Report, LBNL-39098, Lawrence Berkeley National Laboratory. July 1996.
- [Wur95] E. Wurtz. *Modélisation tridimensionnelle des transferts thermiques et aérauliques dans le bâtiment en environnement orienté objet*. PhD thesis, Ecole Nationale des Ponts Chaussées, December 1995.
- [KK82] Kast and Klan. *Auslegung und prüfung von fubbodenheizungen (Interpretation and test of floor heating)*. VDI-Berichte 464: 39-49. 1982.

List of symbols

Bu : plume width [m]
 C : pressure power-law coefficient [$\text{m.s}^{-1}.\text{Pa}^{-n}$]
 C_{conv} : convective heat transfer coefficient [$\text{W.m}^{-2}\text{K}^{-n_{conv}-1}$]
 C_j : empirical coefficient
 C_p : specific heat of air at constant pressure [$\text{J.kg}^{-1}.\text{K}^{-1}$]
 D_j : specie diffusion coefficient [$\text{kg}.$]
 E_0 : entrainment factor
 g : gravitational acceleration [$\text{m.kg}^{-1}.\text{s}^{-2}$]
 h : cell (or interface) height [m]
 $h(x)$: jet width [m]
 h_{conv} : convective heat transfer coefficient [$\text{W.m}^{-2}\text{K}^{-1}$]
 h_0 : diffuser height [m]
 K_v : empirical coefficient
 L : cell (or interface) length [m]
 w : cell (or interface) width [m]
 n : pressure relationship exponent
 n_{conv} : convective heat transfer exponent
 P : pressure [Pa]
 P_{elec} : heater electrical power [W]
 P_{nom} : heater nominal power [W]
 q_m : mass air flow rate [kg.s^{-1}]
 q_{source} : mass air flow rate from sources [kg.s^{-1}]
 q_{sink} : mass air flow rate to sinks [kg.s^{-1}]
 $Q_{mj}(x)$: maximum jet air flow rate [kg.s^{-1}]
 Q_{mj0} : air flow rate in the diffuser [kg.s^{-1}]
 r : air molar constant [$\text{m}^2.\text{s}^{-2}\text{K}^{-1}$]
 S : surface area [m^2]
 SF_i : fictitious surface [m^2]
 St : Stanton number
 T : temperature [K]
 T_a : average room air temperature [C]
 T_{Fi} : fictitious surface temperature [C]
 T_w : wall surface temperature [C]
 Um : maximum mean plume velocity in a cross section [m.s^{-1}]
 $U_{mj}(x)$: maximum jet velocity [m.s^{-1}]
 U_{mj0} : velocity in the diffuser [m.s^{-1}]
 X_j : specie density ratio [$\text{kg}_{specie}.\text{kg}^{-1}_{air}$]
 z : altitude [m]
 z_0 : altitude of the beginning of the plume [m]
 z_n : altitude of the neutral axis [m]
 β : coefficient of thermal expansion [K^{-1}]
 $\Delta T_{s_{max}}$: maximum temperature difference between front face of heater and ambient

e : emissivity
 e_{Fi} : fictitious surface emissivity
 l : ratio between thermal and dynamic plume width
 r : air density [kg.m^{-3}]
 s : Stefan-Boltzmann constant
 j : heat flux [W.m^{-2}]
 j_{conv} : convective heat flux [W.m^{-2}]
 j_{rad} : net radiative heat flux [W.m^{-2}]
 F : heat flux [W]
 F_{conv} : convective heat loss [W]
 F_{rad} : net radiative heat loss [W]
 F_{source} : heat source power [W]
 F_{sink} : heat sink power [W]

Subscripts :

$back_face$: back face of the heater
 $front_face$: front face of the heater
 i : surface i or cell i
 $wall$: wall surface