CFD-FREE, EFFICIENT, MICRO INDOOR CLIMATE PREDICTION IN BUILDINGS

Lars Eriksson[1], Grigori Grozman[1], Pavel Grozman[1], Per Sahlin[1], Mette Havgaard Vorre[2], and Lars Ålenius[1]
[1]EQUA Simulation AB, Solna, Sweden
[2]Danish Building Research Institute, Aalborg University, Hørsholm, Denmark

ABSTRACT

Today, the only accessible simulation method to gain insight into more detailed room thermal comfort is to use CFD. Unfortunately, CFD is much too cumbersome to apply on a routine basis. To gain information that represents longer periods (such as years) and that includes radiation and thermal mass effects, execution time becomes prohibitively long. Another fundamental problem with CFD is that significant work must be spent on the development of a suitable computational grid. This paper investigates the possibility of obtaining sufficiently accurate design information by improving the accuracy of a traditional well-mixed building simulation zone model. A Modelica zone model has been developed that is enhanced by three means: (1) view factors are computed for arbitrary, also self-obstructing, zone geometries, (2) vertical gradients are computed by a multi-node 1D model, and (3) jet and plume flow element theory is applied to track individual jets from salient heat or mass sources. The main design decisions are presented along with some preliminary computational results.

INTRODUCTION

The main variables for determining thermal comfort are, in rough order of importance: direct sunlight, air temperature, radiant temperature, air velocity, and air moisture. Correct design decisions about energy vs. comfort require information about the simultaneous magnitude of these quantities during the whole season when heating or cooling systems are in operation.

In a well-designed room, air velocities in the occupied zone will remain below a level where draft can be sensed, approximately 0.1 m/s. From a design perspective, there is no primary need to describe the full flow field of the room. It is sufficient to ensure that high velocities are unlikely to occur in the occupied zone. In atria, other tall rooms, or in rooms with displacement ventilation, the vertical temperature distribution can also have a significant impact on energy use and comfort. For non-industrial applications, contaminant levels are primarily related to long-term health effects rather than comfort, and their variation within a room will seldom be a key design issue.

Given the enormous possibilities to predict flow fields that CFD has brought to numerous domains, it is only natural that much of the scientific discussion on indoor climate and building energy use also has centered on CFD. However, as a means of economically assessing draft risk and vertical temperature gradients, CFD has serious shortcomings. Key to predictions of draft risk in an indoor environment is the understanding of jets and plumes, i.e. regions of relatively high velocities in a uniform direction in an otherwise almost stagnant fluid. If jets penetrate the occupied region in unintended ways, there will be discomfort.

A turbulent flow, such as that of most jets and plumes, features a chaotic system of vortices over a wide range in both time and space. Any practical CFD method must therefore provide a turbulence model that allows some time average of the flow to be predicted instead of the real fluctuating flow. The global effects of a turbulent jet or plume flowing into still air will, at a fundamental level, depend on this complex system of vortices. These effects can, in principle, be computed by a numerical solution of the Navier-Stokes equations, using a fine grid and some suitable empirical turbulence model, or they can be described directly by semi-empirical analytical expressions – flow elements.

Since the early nineties, the concept of zonal models has been investigated by several researchers to overcome the computational burden of CFD for building simulation (Inard, Bouia, & Dalicieux 1996) (Haghighat, Li, & Megri 2001) (Stewart, Ren 2006) (Song et al. 2008) (Villi, Pasut, & De Carli 2009) (Norrefeldt, Nouidui, & Grün 2010). Basically, they aim at obtaining a sufficiently detailed flow field but with a much coarser grid than is applied in a normal CFD study. Several researchers have utilized some form of explicit jet models in their zonal models, and a common concept is to ask the user to divide the domain into suitable subzones to capture the main flow pattern. In spite of these efforts, (Mora, Gadgil, & Wurtz 2003) conclude that coarse grid CFD still delivers more cost-effective results than this class of zonal models.

None of the proposed zonal models has, to the best of our knowledge, been successfully implemented and appreciated in a widely used whole-building simulation package. One possible reason is the static nature of the flow field that is required in order to optimize the subzone structure with respect to the flow – a requirement for most zonal models. However, many of the most relevant flow elements are in fact far from static. The centerline path will be entirely different depending on the actual air flow and temperature difference. The risk of a jet detaching from the ceiling and "falling down" into the occupied zone under some conditions is indeed one of the most relevant things to study, and it can not be done if the model itself prescribes a static flow path.

Flow element theory is based on a combination of theoretical fluid dynamics and empirical experiments and observations. In (Nielsen 2007) flow elements, CFD and full-scale experiments are discussed as complementary methods for the design of room air distribution systems. Nielsen argues that flow elements are well suited when ventilation is either based on mixing or displacement strategies. On the other hand, some air distribution systems cannot today be adequately described by flow elements, (such as a textile ceiling diffusor inlet). In these situations, CFD or full scale experiments are preferable according to (Nielsen 2007).

The description of flow elements is continuously growing, covering more flows and interactions; (Cao, Ruponen & Kurnitski 2010) has, for example, recently contributed to the description of flow elements describing the velocity distribution when a plane jet collides with a corner.

Air terminal manufacturers make measurements of the velocity distribution from their devices. These results can be used in simulations with flow elements to compute initial flow element boundary conditions.

The purpose of this work is to develop a new method and framework for computation of flow element propagation in rooms with arbitrary polygon-delimited geometry and to apply this to the simplest possible "zonal" model: a vertical division of the zone air into layers. The implementation of the model into a widely used whole building simulator, IDA Indoor Climate and Energy (IDA ICE), will enable a thorough evaluation of the range of accuracy and practical applicability of the model by independent researchers. As with any IDA ICE model, the model is implemented as open, equation-based source code, enabling independent experimentation with variations.

In the next section, the basic properties of a new zone model will be introduced. After this, the main equations of the model will be presented; finally, the paper concludes with some preliminary computational results, the present state of the project, and future plans.

DESIGN DECISIONS

The new zone model has been developed in Modelica (www.modelica.org) for use in IDA ICE. It has three main characteristics, which separate it from previous state of the art:

1. A code for computation of view factors for arbitrary polygon delimited zone geometries with obstructions is implemented.
2. A simplified model for estimation of vertical temperature gradient has been implemented. Essentially, this is the same type of model as is used for temperature prediction in a stratified thermal storage tank.
3. Flow element models have been associated with specific room features such as hot or cold vertical surfaces, mechanical ventilation terminals, occupants, equipment, and flow openings. Within the reach of each flow element, (absolute) air velocity and temperature are given by the flow element model. If two or more flow elements coincide, the most influential model as determined by velocity prevails.

Cartesian grids for field measurement have been defined. Based on these, various comfort measures are computed on the same grid.

Radiation and View factors

As already discussed, correct modelling of radiation is essential to predict thermal comfort. In the new model, the location of the illuminated patch of direct sunlight is computed for each window. Diffuse shortwave and longwave radiation exchange between surfaces is computed by the radiosity method, relying on view factors. For simple box geometries, the matter is straightforward; view factors can easily be computed. However, even for an L-shaped room, a simple algorithm will fail. Not all surfaces will "see" each other. In modern architecture irregular shapes are extremely common.

In the new zone model, a program by Walton (Walton 2002) that computes view factors for arbitrary polygon delimited volumes has been employed.

1D vertical finite difference model

Several authors have proposed and for some situations validated the simplest possible of "zonal models": a vertical division of the zone air into several layers (stacked zones). The Mundt model, based on an approximation of a linear vertical gradient, was implemented in the very first release of IDA Indoor Climate and Energy in 1998. It has shown favorable agreement with real displacement ventilation situations, not only in lab measurements but also by application in many actual projects. Other authors have tested multi node models that also performed well in displacement ventilation situations (Li, Sandberg & Fuchs 1992) (Rees & Haves 2001).

The model selected for this work was first proposed by (Togari, Arai, & Miura 1993). The model has been favorably validated with respect to test cell and full-scale atrium measurements by (Arai, Togari, & Miura 1994) and (Takemasa, Togari, & Arai 1996). The model is based on a 1D vertical finite difference model which is augmented with explicit disturbances in the form of any number of jets, plumes, and wall currents.

Flow elements

All mass in-flows are attributed to individual flow elements that are tracked until they are regarded as dissolved. Out-flows are assumed to have approximately spherical isovelocity surfaces. Individual thermal plumes are generated at each distinct heat source and also where the direct sunlight from a window hits a surface. Convective flows that are generated by heat transfer at the main surfaces are also treated by special wall current models.

For the testing of the proposed approach, focus has been put on wall currents and on 3D jets and plumes. The progress of each 3D flow element is integrated along its trajectory, with the following conditions for shifting to a new segment:

1. The flow element is leaving a layer of constant temperature
2. The flow element is impinging on a surface
3. The flow element is detaching from a surface
4. The centerline velocity of the flow element is below 0.1 m/s (in which case the flow element is terminated)

In reality, flow elements need to progress a bit until the characteristic, self-similar flow has been established. For a round jet, the flow has been established approximately 8 diameters (of the source) downstream. At this stage in this work, we disregard the different flow type during the establishment phase. The work will be extended to include radial jets, gravity currents, and other types of analytically predictable flows once the initial validation has succeeded.

Figure 1 shows a visualization of the temperature on a plane around and within a flow element from an open roof window.

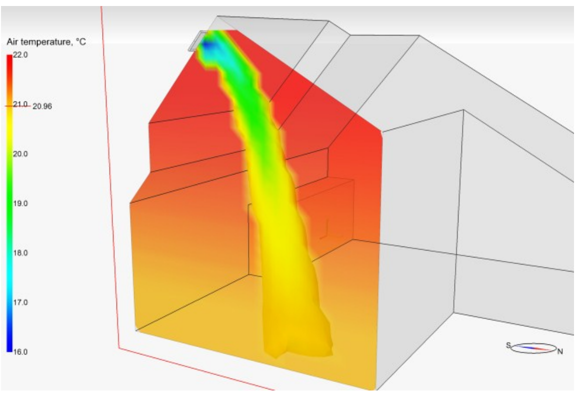

Figure 1. Flow element visualization

THEORY

In this section, a mathematical description of the core of the model is given.

Mass balance

The zone is discretized in horizontal layers starting from the floor. Flows are defined as positive in the upward direction and from flow elements to layers. To simplify the notation, we include leaks, openings, and wall currents in what is referred to as flow elements below.

$$\dot{m}_{lay\_i} = \dot{m}_{lay\_i-1} + \dot{m}_{flow\_i}, \quad i = 1, nLay, \quad \dot{m}_{lay\_0} = 0$$

$$0 = \sum_{i=1}^{nLay} \dot{m}_{flow\_i} = \dot{m}_{lay\_nLay}$$

where

i	=	layer number, from floor upward
$\dot{m}_{lay\_i}$	=	air mass flow between layer i and $i+1$ (kg/s)
$\dot{m}_{flow\_j\_i}$	=	$\dot{m}_{flow\_j\rightarrow i} - \dot{m}_{flow\_j\leftarrow i} =$ net air mass flow between flow element j and layer i (kg/s)
$\dot{m}_{flow\_j\rightarrow i}$	=	air mass flow from flow element j to layer i (kg/s)
$\dot{m}_{flow\_j\leftarrow i}$	=	air mass flow from layer i to flow element j (kg/s)
$nLay$	=	number of layers
$nFlow$	=	number of flow elements.

Heat balance

The heat transport is given by equation

$$\rho V_i \left((c_{pAir} + c_{pVap} W_i) \frac{dT_i}{dt} + h_{pVap} \frac{dW_i}{dt} \right) =$$

$$= \Delta Q_{cond\_i} + \Delta Q_{trans\_i} + \Delta Q_{inv\_i} + Q_{flow\_i} + Q_{other\_i}$$

Where

$$Q_{cond\_i} = \lambda_{air} A_{top\_i} 2 \frac{T_i - T_{i+1}}{h_i + h_{i+1}}, \quad i = 1, nLay - 1$$

$$Q_{cond\_0} = 0, \quad Q_{cond\_nLay} = 0$$

$$Q_{trans\_i} = \max(0, \dot{m}_{lay\_i}) E_i + \min(0, \dot{m}_{lay\_i}) E_{i+1}$$

$$\dot{m}_{inv\_i} = 4.097 \cdot 10^{-5} \rho L_{turb}^2 A_{top\_i} \left(2 \frac{\max(0, T_i - T_{i+1})^{1/3}}{h_i + h_{i+1}} \right)^{3/2},$$

$$i = 1, nLay - 1$$

$$Q_{inv\_i} = \dot{m}_{inv\_i}(E_i - E_{i+1}), \quad i = 1, nLay - 1,$$

$$Q_{inv\_0} = 0, \quad Q_{inv\_nLay} = 0$$

$$\Delta Q_i = Q_{i-1} - Q_i$$

and

V_i	=	volume of layer i (˚C)
T_i	=	air temperature in layer i (˚C)
$T_{flow\_j\_i}$	=	air temperature in flow element j in layer i (˚C)
T_{floor}	=	floor temperature (˚C)
$T_{ceiling}$	=	ceiling temperature (˚C)

W_i	=	humidity ratio in layer i (kg/kg)
E_i	=	$c_{pAir}\, T_i + W_i\,(h_{pVap} + c_{pVap}\, T_i\,)$ = air enthalpy in layer i (J/kg)
$E_{flow\_j\_i}$	=	$c_{pAir}\, T_{flow\_j\_i} + W_{flow\_j\_i}\,(h_{pVap} + c_{pVap}T_{flow\_j\_i})$ = enthalpy in flow element j in layer i (J/kg)
ρ	=	density of air (kg/m^3)
c_{pAir}	=	specific heat of air (J/kg/°C)
c_{pVap}	=	water vapor specific heat (J/kg/°C)
h_{pVap}	=	water vaporization heat (J/kg)
λ_{air}	=	conductivity in air (W/m/°C)
h_{air}	=	heat transfer coefficient (W/ m^2/°C)
L_{Turb}	=	empirical parameter for temperature inversion (m)
Q_{cond_i}	=	heat from conduction between layers i and i+1 (W)
Q_{trans_i}	=	heat through mass transport between layers i and i+1 (W)
$\dot{m}_{inv\_i}$	=	inversion mass flow between layers i and i+1 (kg/s)
Q_{inv_i}	=	heat from inversion between layers i and i+1 (W)
Q_{flow_i}	=	net heat flow between flow elements and layer i (W)
$Q_{flow\_j\rightarrow i}$	=	heat flow from flow element j to layer i (W)
Q_{other_i}	=	local heat sources/sinks, heat exchange with walls, etc. in layer i (W)
h_i	=	height of layer i (m)
$A_{top\ i}$	=	cross section area at top of layer i (m^2)
$A_{bot\ i}$	=	cross section area at bottom of layer i (m^2).

Vapor balance

The vapor transport is given by equation

$$\rho V_i \frac{dW_i}{dt} = \Delta F_{trans\_i} + \Delta F_{inv\_i} + F_{flow\_i} + F_{other\_i}$$

where

$$F_{trans\_i} = \max(0, \dot{m}_{lay\_i})W_i + \min(0, \dot{m}_{lay\_i})W_{i+1}$$

$$F_{inv\_i} = \dot{m}_{inv\_i}(W_i - W_{i+1}), \quad i = 1, nLay-1,$$

$$F_{inv\_0} = 0, \quad F_{inv\_nLay} = 0$$

$$\Delta F_i = F_{i-1} - F_i$$

$$F_{flow\_i} = \sum_{j=1}^{nFlow} \left(\dot{m}_{flow\_j\rightarrow i} W_{flow\_j\_i} - \dot{m}_{flow\_j\leftarrow i} W_i \right)$$

and

W_i	=	humidity ratio in layer i (kg/kg)
$W_{flow\_j\_i}$	=	humidity ratio in flow element j in layer i (kg/kg)
ρ	=	density of air (kg/m^3)
$F_{trans\ i}$	=	vapor transport between layers i and i+1 (kg/s)
$\dot{m}_{inv\_i}$	=	inversion mass flow between layers i and i+1 (kg/s)
F_{inv_i}	=	vapor flow from inversion between layers i and i+1 (kg/s)
F_{flow_i}	=	net vapor flow between flow elements and layer i (kg/s)
$F_{other\ i}$	=	local vapor sources/sinks, etc (kg/s).

CO_2 balance

The CO_2 transport is given by equation

$$\rho V_i \frac{dX_i}{dt} = \Delta F_{trans\_i} + \Delta F_{inv\_i} + F_{flow\_i} + F_{other\_i}$$

where

$$F_{trans\_i} = \max(0, \dot{m}_{lay\_i})X_i + \min(0, \dot{m}_{lay\_i})X_{i+1}$$

$$F_{inv\_i} = \dot{m}_{inv\_i}(X_i - X_{i+1}), \quad i = 1, nLay-1,$$

$$F_{inv\_0} = 0, \quad F_{inv\_nLay} = 0$$

$$\Delta F_i = F_{i-1} - F_i$$

$$F_{flow\_i} = \sum_{j=1}^{nFlow} \left(\dot{m}_{flow\_j\rightarrow i} X_{flow\_j\_i} - \dot{m}_{flow\_j\leftarrow i} X_i \right)$$

and

X_i	=	fraction CO_2 in layer i (mg/kg)
$X_{flow\_j\_i}$	=	fraction CO_2 in flow element j in layer i (mg/kg)
ρ	=	density of air in layer i (kg/m^3)
$F_{trans\ i}$	=	CO_2 transport between layers i and i+1 (mg/s)
$\dot{m}_{inv\_i}$	=	inversion mass flow between layers i and i+1 (kg/s)
F_{inv_i}	=	CO_2 flow from inversion between layers i and i+1 (mg/s)
F_{flow_i}	=	net CO_2 flow between flow elements and layer i (mg/s)
$F_{other\ i}$	=	local CO_2 sources/sinks, etc (mg/s).

Flow elements

The trajectory of each flow element is computed by numerically integrating the following differential equations (generalized from Etheridge & Sandberg 1996) with appropriate initial conditions

$$\frac{dq_p}{ds} = \alpha\sqrt{8\pi m}$$

$$\frac{dB}{ds} = -g\beta q_p \frac{dT_\infty}{ds}$$

$$\frac{dm}{ds} = \left| \mathbf{F}_{\|} \right|$$

$$\frac{d\Theta}{ds} = \frac{1}{R}$$

$$\frac{dx_c}{ds} = \cos(\Theta)$$

$$\frac{dy_c}{ds} = \sin(\Theta)$$

where

$$\mathbf{F} = \mathbf{F}_B + \mathbf{F}_C = \mathbf{F}_{||} + \mathbf{F}_{\perp}$$

$$\mathbf{F}_B = \frac{B}{u_c}\frac{I_1}{I_2}\mathbf{e}_y$$

$$u_c = |\mathbf{u}| = \frac{m}{q_p}\frac{I_1}{I_2}$$

$$R = \frac{m}{|\mathbf{F}_\perp|}$$

where

s = coordinate along trajectory from jet origin (m)
Θ = deviation angle relative horizontal plane (rad)
$\mathbf{e}_y$ = unit vector vertically downward, direction of buoyancy (-)
x_c = center line horizontal position from jet origin (m)
y_c = center line vertical position from jet origin (m)
u_c = $|\mathbf{u}|$ = center line velocity (m/s)
$\mathbf{u}$ = vector center line velocity (m/s)
q_p = volume flow (m^3/s)
m = specific momentum flux (m^4/s^2)
B = specific buoyancy flux (m^4/s^3) = $2.8 \times 10^{-5} Q_{source}$ (for a plume originating from a heat source)
$\mathbf{F}$ = total force per unit mass and unit length (m^3/s^2)
$\mathbf{F}_B$ = buoyancy force per unit mass and unit length (m^3/s^2)
$\mathbf{F}_C$ = external body force per unit mass and unit length (m^3/s^2)
$\mathbf{F}_\parallel$ = force per unit mass and unit length parallel to $\mathbf{u}$ (m^3/s^2)
$\mathbf{F}_\perp$ = force per unit mass and unit length perpendicular to $\mathbf{u}$ (m^3/s^2)
T_∞ = ambient temperature (layer temperature) (˚C)
α = coefficient of entrainment (-)
β = thermal expansion coefficient = $1/T$ for a perfect gas (1/K)
g = gravitational acceleration (m/s^2)
I_i = jet profile constants, i = 1,3 (-)
R = deviation radius (m)
ΔT = difference between centerline temperature and ambient temperature = $\frac{B}{\beta g q_p}\frac{I_1}{I_3}$ (˚C).

When the trajectory of the jet is calculated, one obtains a sequence of known volume flows. Observing how this volume flow changes as a function of zone height, one can deduce layer volume exchange $q_{p\_k \to i}$, where $k \to i$ denotes the non-negative flow from layer k to layer i through the flow element. Hence if j is the number of the flow:

$$\dot{m}_{flow\_j \to i} = \rho \cdot \sum_{k=1}^{nLay} (q_{p\_k \to i}) + \dot{m}_{in\_j} D_{j\_i}$$

$$\dot{m}_{flow\_j \leftarrow i} = \rho \cdot \sum_{k=1}^{nLay} (q_{p\_i \to k})$$

$$Q_{flow\_j \to i} = \frac{\rho \cdot \sum_{k=1}^{nLay} (q_{p\_k \to i} \cdot E_k) + Q_{in\_j} D_{j\_i}}{\dot{m}_{flow\_j \to i}}$$

$$W_{flow\_j\_i} = \frac{\rho \cdot \sum_{k=1}^{nLay} (q_{p\_k \to i} \cdot W_k) + \dot{W}_{in\_j} D_{j\_i}}{\dot{m}_{flow\_j \to i}}$$

$$X_{flow\_j\_i} = \frac{\rho \cdot \sum_{k=1}^{nLay} (q_{p\_k \to i} \cdot X_k) + \dot{X}_{in\_j} D_{j\_i}}{\dot{m}_{flow\_j \to i}}$$

where

$\dot{m}_{in\_j}$ = mass flow inserted into zone by flow element j (kg/s)

D_{j_i} = fraction of the inserted mass flow delivered to layer i by flow element j (-)

Q_{in_j} = heat inserted into zone by flow element j (W)

$\dot{W}_{in\_j}$ = humidity flow inserted into zone by flow element j (kg/s)

$\dot{X}_{in\_j}$ = CO_2 flow inserted into zone by flow element j (mg/s).

Wall currents

If the wall temperature is less than the mean air temperature in the zone, the air is chilled and flows down along the wall in thin layers creating wall currents. Here, the model by (Togari, Arai, & Miura 1993) has been adopted. The volume flow from air layer i to the corresponding wall current and its temperature is computed as

$$V_{out\_i} = \frac{4 \cdot h_{air} \cdot A_{wall\_i}}{\rho \cdot c_{pAir}}$$

$$T_{out\_i} = 0.75 \cdot T_i + 0.25 \cdot T_{wall}$$

where

V_{out_i} = volume flow from air layer i to wall current (m^3/s)

T_{out_i} = temperature of air entering the wall current from layer i (˚C)

T_i = air temperature in layer i (˚C)

T_{wall} = wall temperature (˚C)

h_{air} = film coefficient between wall and air layer (W/m^2/˚C)

A_{wall_i} = area of wall in layer i (m^2)

ρ = density of air (kg/m^3)

c_{pAir} = specific heat of air (J/kg/˚C).

The volume flow in wall current from layer i+1 and air flow from air layer i mix yielding

$$V_{m\_i} = V_{md\_i+1} + V_{out\_i}$$

and

$$T_{m\_i} = (V_{md\_i+1} T_{m\_i+1} + V_{out\_i} T_{out\_i}) / V_{m\_i}$$

$$W_{m\_i} = (V_{md\_i+1} W_{m\_i+1} + V_{out\_i} W_i) / V_{m\_i}$$

$$X_{m\_i} = (V_{md\_i+1} X_{m\_i+1} + V_{out\_i} X_i) / V_{m\_i}$$

where

V_{md_i+1}	=	air volume flow of wall current from layer i+1 to layer i (m^3/s)
V_{m_i}	=	air volume flow of wall current at layer i (m^3/s)
T_{m_i}	=	temperature of wall current at layer i (˚C)
W_{m_i}	=	humidity ratio of wall current at layer i (kg/kg)
X_{m_i}	=	fraction CO_2 of wall current at layer i (mg/kg)
W_i	=	humidity ratio in layer i (kg/kg)
X_i	=	fraction CO_2 in layer i (mg/kg).

Some of the air in V_{md_i} continues down to layer i-1 and some returns to air layer i. The splitting is done using a coefficient p_i.

$$V_{md\_i} = p_i \cdot V_{m\_i}$$

$$V_{in\_i} = (1 - p_i) \cdot V_{m\_i}$$

$$p_i = \begin{cases} 0 & \text{if } T_{m\_i} \geq T_i \\ 1 & \text{if } T_{i-1} \geq T_i \\ \dfrac{T_i - T_{m\_i}}{T_i - T_{i-1}} & \text{otherwise} \end{cases}$$

where

V_{in_i} = air volume flow from wall current to air layer i (m^3/s).

If the wall temperature is higher than the mean air temperature in the zone, then the current flows up along the wall. Such a flow is modeled in a similar way as described above.

Hence, the influence of the wall current to the zone is:

$$\dot{m}_{flow\_j \to i} = \rho \cdot V_{in\_i}$$

$$\dot{m}_{flow\_j \leftarrow i} = \rho \cdot V_{iout\_i}$$

$$Q_{flow\_j \to i} = \rho \cdot V_{in\_i} \cdot E_{m\_i}$$

$$W_{flow\_j\_i} = W_{m\_i}$$

$$X_{flow\_j\_i} = X_{m\_i}$$

where

E_{m_i}	=	$c_{pAir} T_{m\_i} + W_{m\_i} (h_{pVap} + c_{pVap} T_{m\_i})$ = enthalpy of wall current in layer i (J/kg)
c_{pVap}	=	water vapor specific heat (J/kg/˚C)
h_{pVap}	=	water vaporization heat (J/kg).

INITIAL RESULTS

So far, two sets of measurements have been treated using the new model for testing its predictive capabilities with respect to wall currents, plumes, and long wave radiation.

MINIBAT test cell

The wall current heat transfer properties of the new zone model have been initially tested with respect to measurements by (Inard, Bouia, & Dalicieux 1996) in the (3.1x3.1x2.5m) MINIBAT test cell (Allard et al. 1987). Four cases have been compared, where surfaces were maintained at given temperatures according to Table 1. No solar radiation or ventilation was applied in these experiments. Five vertical layers were used; no study has been conducted to find an optimal number.

Table 1. Measured mean surface temperatures for the MINIBAT test cases

	S	N	E	W	Ceil	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

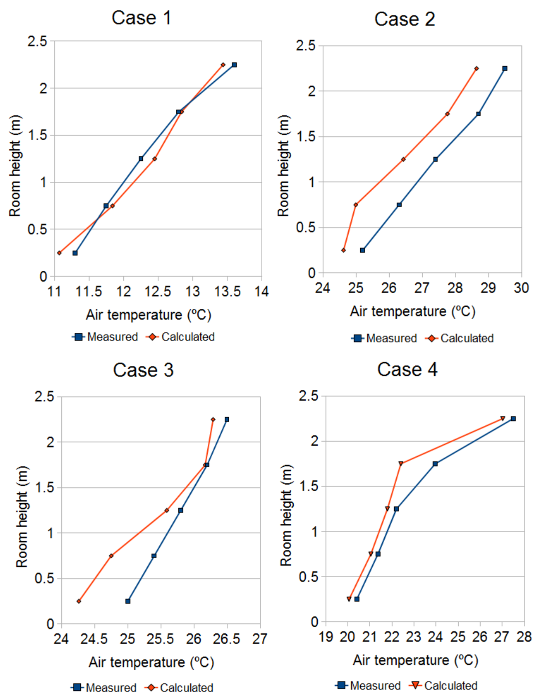

Figure 2. Measurements and calculated results for the MINIBAT test cell

Generally, the average room temperature is slightly under predicted. Some experiments with the model reveal that this is a direct consequence of the type of correlations used for film coefficients. The results of Figure 2 were obtained using simple fixed coefficients.

Film coefficients aside, the model performs rather well in the prediction of vertical temperature gradient.

Gävle test cell

To evaluate the model with respect to displacement ventilation, long wave radiation, and thermal plumes, a test carried out at the Gävle laboratory of the National Swedish Institute for Building Research was used (Li, Sandberg, and Fuchs, 1992 & 1993). The (4.2x3.6x2.75m) room was ventilated by a (0.50x0.45m) low velocity terminal located at floor level. Air is extracted at ceiling level. A cubic, porous heat source (0.4x0.3x0.2m) at 0.1m from the floor and 2.7m from the supply air terminal provides a convective heat load. Wall and ceiling U-values are 0.36 W/m^2K, except for one wall (0.15 W/m^2K) towards guard zones with given temperatures (generally 23°C). The floor temperature outside the test cell was not measured; for the purpose of this study, it was assumed to be 23°C. Thirty vertical temperature layers were modelled to match the temperature sensors of the experiment. Walls were modelled as a single surface with uniform temperature. No experiments with other layer refinement or vertically separated wall temperatures were carried out.

Table 2. Some key characteristics of Gävle test cases

	Sup. air flow [ACH]	Sup. air temp. [°C]	Heat [W]	Wall emiss.
Case A2	3	18.0	300	0.1
Case B2	2	19.2	300	0.9
Case B3	3	18.0	300	0.9
Case B4	3	18.0	450	0.9

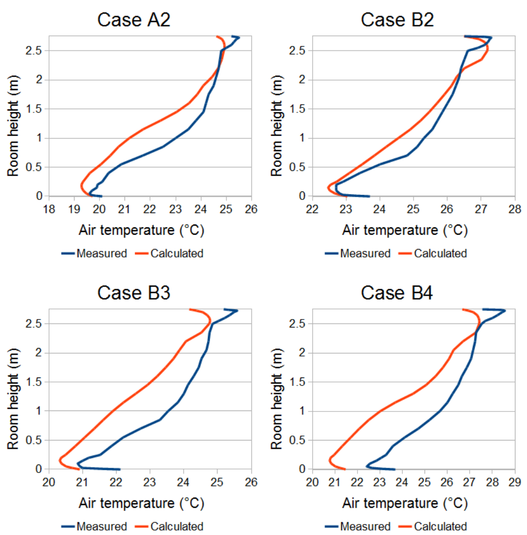

Figure 3. Measurements and calculated results for the Gävle test cell

Also here (Figure 3), there is a general under prediction of room air temperature. The amount of air flow into the zone is not very accurately specified in the account of the measurement (only given as a single digit ACH), i.e. there may be some problem with the overall heat balance of the test cell. However, there may also be some problem with applying traditional, well mixed zone film coefficients for this type of model. This is an area that needs further investigation.

The prediction of the gradient is quite reassuring.

PRESENT STATE

Currently, the model is operational in an experimental version of IDA ICE with a rather primitive user interface. The initial testing has been positive in terms of results, computation time, and robustness. Therefore, the plan is to integrate it fully into the tool. A beta version can be expected by Q3 2012.

Fundamental work still remains in a number of areas:

1. A 3D model for the Coanda effect (jet wall attraction) will be proposed; present models in the literature are limited to simple geometries.
2. Models and methods for 3D flow element transitions will be further developed, e.g., from a free to a wall jet, passage of corners, etc.
3. Further flow elements will be developed.
4. Better models for internal film coefficients can be developed. Traditional correlations are intended to be used with respect to average (mixed) air temperatures, and here we have considerably more information that could be utilized, both in terms of local air temperature near the wall (floor or ceiling) and also in terms of velocity estimates near each surface.

It should be recognized that the new model is also useful with a single (well mixed) air layer for computation of comfort measures that include both shortwave and longwave local radiation.

CONCLUSION AND OUTLOOK

A new type of zone model has been implemented in Modelica in the whole-building, full year building performance simulation program IDA Indoor Climate and Energy. Initial tests look favorable.

In the next stage of this work, validation of the stratification properties of more complex measured cases will be investigated and a description of its range of applicability will be formulated. The authors welcome cooperation with researchers who would like to investigate, validate, and improve the model further.

REFERENCES

Allard. F.. Brau. J.. Inard. C. & Pallier. J. 1987. "Thermal experiments of full-scale dwelling cells in artificial climatic conditions". *Energy and Buildings*. vol. 10. no. 1. pp. 49-58.

Arai. Y., Togari. S., and Miura. K. 1994. "Unsteady-state Thermal Analysis of a Large Space with

Vertical Temperature Distribution” *ASHRAE Trans.*, 100 _part 1_, pp. 396–411

Bring. A.. Sahlin. P. & Vuolle. M. 1999. *Models for Building Indoor Climate and Energy Simulation*. Dept. of Building Sciences.

Cao. G.. Ruponen. M. & Kurnitski. J. 2010. "Experimental investigation of the velocity distribution of the attached plane jet after impingement with the corner in a high room". *Energy and Buildings.* vol. 42. no. 6. pp. 935-944.

Etheridge. D.W. & Sandberg. M. 1996. *Building ventilation: theory and measurement.* John Wiley & Sons.

Griffith. B.T. 2002. *Incorporating nodal and zonal room air models into building energy calculation procedures*. Massachusetts Institute of Technology.

Haghighat. F.. Li. Y. & Megri. A.C. 2001. "Development and validation of a zonal model--POMA". *Building and Environment.* vol. 36. no. 9. pp. 1039-1047.

Heiselberg. P. 2000. "Characteristics of buoyant flow from open windows in naturally ventilated rooms". *AIR DISTRUBUTION IN ROOMS Ventilation for Health and Sustainable Environment Volume II.* pp. 825.

Heiselberg. P.. Bjørn. E. & Nielsen. P.V. 2002. "Impact of open windows on room air flow and thermal comfort". *International Journal of Ventilation.* vol. 1. no. 2. pp. 91-100.

Heiselberg. P.. Svidt. K. & Nielsen. P.V. 2001. "Characteristics of airflow from open windows". *Building and Environment.* vol. 36. no. 7. pp. 859-869.

Inard. C. Bouia. H. & Dalicieux. P. 1996. "Prediction of air temperature distribution in buildings with a zonal model". *Energy and Buildings.* vol. 24. no. 2. pp. 125-132.

Li, Y., Sandberg, M., & Fuchs, L. 1993. "Effects of thermal radiation on airflow with displacement ventilation: an experimental investigation", *Energy and Buildings*, vol. 19. no. 4, pp. 263-274.

Li, Y., Sandberg, M., & Fuchs, L. 1992. "Vertical temperature profiles in rooms ventilated by displacement: full-scale measurement and nodal modelling". *Indoor Air*. vol. 2, no. 4, pp. 225–243.

Mora. L.. Gadgil. A. & Wurtz. E. 2003. "Comparing zonal and CFD model predictions of isothermal indoor airflows to experimental data". *Indoor Air.* vol. 13. no. 2. pp. 77-85.

Mundt. E. 1996. *The Performance of Displacement Ventilation Systems -- Experimental and Theoretical Studies*. Royal Institute of Technology.

Nielsen. P.V. 2007. "Analysis and design of room air distribution systems". *HVAC&R Research.* vol. 13. no. 6. pp. 987-997.

Nielsen. P.V. 1991. *Models for the prediction of room air distribution.* Instituttet for Bygningsteknik. Aalborg Universitetscenter.

Nielsen. P.V. 1993. *Displacement ventilation - theory and design.* Department of Building Technology and Structural Engineering. Aalborg University. Aalborg. Denmark.

Nielsen. P.V. 1994. "Air Distribution in Rooms - Research and Design Methods". *ROOMVENT '94. Fourth International Conference on Air Distribution in Rooms*.

Norrefeldt. V.. Nouidui. T. & Grün. G. 2010. "Development of an isothermal 2D zonal air volume model with impulse conservation".

Rees. S.J. & Haves. P. 2001. "A nodal model for displacement ventilation and chilled ceiling systems in office spaces". *Building and Environment.* vol. 36. no. 6. pp. 753-762.

Song. F.. Zhao. B.. Yang. X.. Jiang. Y.. Gopal. V.. Dobbs. G. & Sahm. M. 2008. "A new approach on zonal modeling of indoor environment with mechanical ventilation". *Building and Environment.* vol. 43. no. 3. pp. 278-286.

Stewart. J. & Ren. Z. 2006. "COwZ--A subzonal indoor airflow, temperature and contaminant dispersion model". *Building and Environment.* vol. 41. no. 12. pp. 1631-1648.

Svidt. K.. Heiselberg. P.K. & Nielsen. P.V. 2000. "Characterization of Airflow from a Bottom Hung Window for Natural Ventilation". July 9-12. 2000. pp. 755.

Takemasa. Y., Togari. S., and Arai. Y. 1996. “Application of an unsteady-state model for predicting vertical temperature distribution to an existing atrium”. *ASHRAE Transactions*, 102, (1)

Togari, S., Arai, Y., and Miura, K. 1993. "A simplified model for predicting vertical temperature distribution in a large space". *ASHRAE Transactions*, 99(part 1): 84–99

Villi. G.. Pasut. W. & De Carli. M. 2009. "Computational aspects of modelling different strategies for kitchen ventilation: A comparison between the multi-zone approach and CFD modelling with reference to predicted indoor pollutant concentrations". *Building Simulation 2009 Eleventh International IBPSA Conference Glasgow. Scotland*. July 27-30.

Walton. G.N. 2002. *Calculation of obstructed view factors by adaptive integration*. National Institute of Standards and Technology, NISTIR 6925.