Modeling Local Hygrothermal Interactions

Paul W.M.H. Steskens, PhD **Hans Janssen, PhD** **Carsten Rode, PhD**

ABSTRACT

Researchers are currently striving to advance possibilities for calculating the integrated phenomena of heat, air, and moisture (HAM) flows in buildings, while including the interactions that take place between the various building materials, components, and room air. Coupled HAM building simulation models have been developed by coupling numerical models that describe the airflow in the rooms of a building with HAM building component models. Computational fluid dynamics (CFD) models have been used to model the local indoor environmental conditions and convective surface transfer coefficients.

As an alternative to the use of CFD models, which are strongly limited by computer capacity, the applicability of subzonal airflow models for transient HAM building simulations has been investigated. This paper presents the modeling of the local indoor environmental conditions and convective surface transfer coefficients, focusing on prediction of the local interior surface heat and moisture transfer coefficients. The research showed that the developed model gives good agreement with the local convective surface transfer coefficients predicted from CFD. The main advantage of the presented subzonal airflow model is that the computational effort is relatively small, while predictions of the local surface transfer coefficients can be relatively accurate.

INTRODUCTION

Heat, air, and moisture flows that are generated inside a building and traverse the enclosure and the flows injected by the heating, ventilation, and air-conditioning (HVAC) system, continuously interact with each other. Airflows, generated by air pressure differences inside and outside buildings, may influence the heat, air, and moisture (HAM) response of the envelope. Resulting moisture deposits in the envelope may negatively affect energy consumption. Moisture from inside and heat and moisture from outside put the envelope's durability at risk.

For the past few decades, the development and professional use of tools to simulate the processes involved in analysis of HAM conditions have increased. Currently, researchers are striving to advance possibilities for calculating the integrated phenomena of HAM flows in buildings, while including the interactions that take place between the various building materials, components, and room air, and the influences due to occupants and HVAC systems. Coupled HAM building simulation models have been developed by coupling a numerical model that describes the airflow in the room of a building with a HAM building component model. Computational fluid dynamics (CFD) models (Steeman 2008) (Mortensen 2007) or subzonal airflow models (Mendonça 2004) have been used to model the local indoor environmental conditions and convective surface transfer coefficients.

The main requirement for successful modeling of the hygrothermal interaction between the building component and the indoor environment is the correct treatment of the interfacial flows at the boundaries (Woloszyn and Rode 2008). Heat, air, and moisture conditions in a building component are dependent on the boundary conditions, i.e., the indoor and outdoor climate conditions. Due to the spatial variability of these climatic conditions, caused by local heat and moisture sources, imperfect mixing and microclimatic effects, the temperature and relative humidity in the neighboring air are

Paul Steskens *is a researcher in the Department of Architecture, Building, and Planning, Eindhoven University of Technology, The Netherlands.* ***Hans Janssen*** *is an associate professor and* ***Carsten Rode*** *is a professor in the Department of Civil Engineering, Technical University of Denmark, Denmark.*

seldom uniform. Similarly, the convective surface heat and moisture transfer coefficients vary in space, due to their strong dependence on, for example, local air velocity and local temperature.

CFD models are capable of predicting the local temperature and relative humidity near a building component as well as the local surface transfer coefficients. However, detailed airflow models cannot easily and quickly solve transient hygrothermal interactions across the boundaries of a HAM building model. In practice, only steady-state simulations of the airflow in a single room at a specific time are feasible. And, since these steady-state calculations are relatively computationally intensive, transient calculations over a longer period of time are currently not possible.

As an alternative to the use of CFD models, which are strongly limited by computer capacity, subzonal airflow models, which describe the airflow in the zone of a building—for example, airflow in a room, in part of the room, or near a building component—can be used. Comprehensive reviews of the literature on subzonal models have been carried out by Teshome and Haghighat (2004) and Megri and Haghighat (2007). The evaluations focused on developments and applications over the last three decades. Applications show that the subzonal modeling approach can be a suitable method to estimate temperature and relative humidity fields in a room with reasonable accuracy. Subzonal models can give a satisfactory estimate of airflow patterns but not highly detailed information on air-speed magnitude. Nevertheless, this approach proved adequate for estimating (annual) indoor thermal comfort (Wurtz et al. 2006). Undetermined is whether the models can provide very detailed information about the local indoor environmental conditions and the local convective surface transfer coefficients in the room compared to CFD.

The objective of this paper is to investigate the applicability of the subzonal model for transient HAM building simulations, focusing on two requirements:

1. The subzonal airflow model should be able to give an accurate prediction of the local temperature and relative humidity near the building component.
2. The model should be capable of providing an accurate prediction of the local convective surface transfer coefficients in the room.

The methodology that has been applied is as follows:

1. A case study that considers the natural convective airflow in a room has been selected for analysis.
2. Subzonal airflow models and surface transfer coefficients models have been simulated in order to predict the local indoor environmental conditions and convective surface transfer coefficients in the room.
3. Numerical results obtained from these models have been compared with both experimental and CFD results.

ANALYSIS AND METHODS

A case for natural convection in a room (Figure 1) is analyzed. Experimental results from the MINIBAT test cell at the Thermal Sciences Centre of Lyon (CETHIL) (Inard et al. 1996) are used. The MINIBAT test cell consists of a 24 m^3 (3.1 × 3.1 × 2.5 m) single volume for which temperature is controlled and kept constant on the faces. The MINIBAT test cell is a room that has been designed in which to study airflow under laboratory conditions. A detailed description of the MINIBAT test cell can be found in Allard et al. (1987). Temperatures of the northern and southern walls, the floor, and the ceiling have been kept constant at 33.0°C, 16.9°C, 26.9°C, and 28.5°C, respectively. The temperature of the western and eastern walls was approximately 27°C. Since a similar surface temperature was applied on the western and eastern walls, the airflow in the center of the room is considered to be two dimensional. The analysis focuses on the symmetry plane. Figure 1 presents a two-dimensional slice of the room along the symmetry plane, and the corresponding geometry and boundary conditions. The relative humidity on both walls was 50% RH, while the floor and ceiling are considered to be vapor tight.

Inard et al. (1996) carried out measurements to investigate the natural convective airflow in the room so as to validate and verify numerical results from a subzonal model. Detailed experimental information regarding the airflow pattern and temperature distribution in the MINIBAT test cell under specific conditions is available. The data have been used for comparison with the results obtained in the present study.

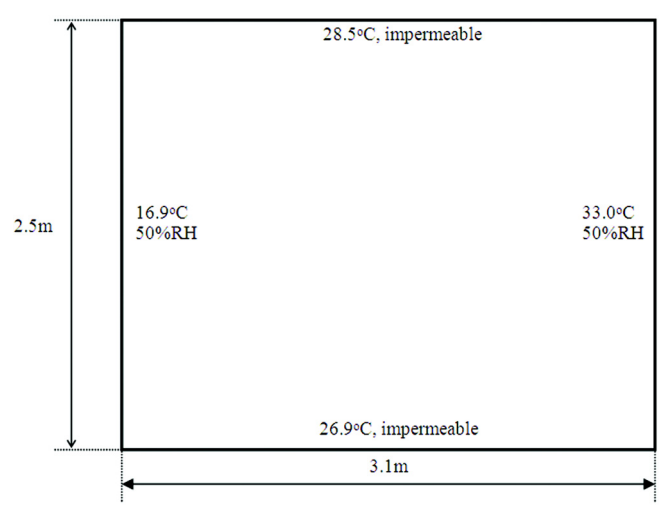

Figure 1 *Geometry and boundary conditions for the MINIBAT case (Inard et al. 1996).*

MODELING

Subzonal Airflow Model

The airflow in the room is modeled using a subzonal airflow model. The room is subdivided into a relatively small number of discrete control volumes, typically less than 1000. Within a subzone, the temperature and relative humidity are considered uniform. In subzonal airflow models, the airflow is governed by a relatively simple set of equations as compared to a CFD model, which solves the Navier-Stokes equations governing the airflow. The steady-state air mass balance of each subzone is governed by Equation 1.

$$\int_{\Omega} \{\rho(\nabla \cdot \underline{u}) + S_{\rho}\} d\Omega = 0 \tag{1}$$

where

Ω = the volume of the subzone, m^3

t = time, s

ρ = fluid's density, $kg \cdot m^{-3}$

$\underline{u}$ = velocity vector, $m \cdot s^{-1}$

S_{ρ} = source term, $kg \cdot m^{-3} \cdot s^{-1}$

Density variations of the air are modeled using the incompressible ideal gas relationship.

The steady-state energy balance in a subzone Ω, with velocity component $\underline{u}$ at temperature T, is expressed by Equation 2.

$$\int_{\Omega} \{\nabla \cdot (\rho c_p T \underline{u}) - \nabla \cdot (\lambda \nabla T) + S_T\} d\Omega = 0 \tag{2}$$

where

c_p = specific heat capacity, $J \cdot kg^{-1} \cdot K^{-1}$

λ = thermal conductivity of the fluid, $W \cdot m^{-1} \cdot K^{-1}$

S_T = represents any heat sources in the fluid, $W \cdot m^{-3}$

Similarly, the steady-state vapor mass balance of each subzone is presented by Equation 3:

$$\int_{\Omega} \{\nabla \cdot (\rho \underline{u} x) - \nabla \cdot (\rho D_v \nabla x) + S_V\} d\Omega = 0 \tag{3}$$

where

x = vapor content per kg dry air, $kg \cdot kg^{-1}$

D_v = vapor diffusivity, $m^2 \cdot s^{-1}$

It should be mentioned that Equation 3 only applies to situations where the temperature of the air is relatively low, assuming that the partial pressure of the dry air is relatively large compared to the partial vapor pressure. Vapor sources in the room are represented by the source term S_v ($kg \cdot m^{-3}$).

With respect to the boundary conditions, the heat transfer to and from the building component to the air perpendicular to the component is represented as a source term (Equation 4).

$$\int_{\Omega} S_T d\Omega = \int_{A} \{n \cdot (\alpha_c (T_{air} - T_{wall}))\} dA \tag{4}$$

where

α_c = convective surface heat transfer coefficient, $W \cdot m^{-2} \cdot K^{-1}$

T_{air}, T_{wall} = air temperature in the center of the control volume and the wall surface temperature, respectively, K

S = surface, corresponding to the height or the width of the control volume, m^2

The boundary conditions for vapor transfer to and from the building component to the air adjacent to the component are represented as a source term:

$$\int_{\Omega} S_M d\Omega = \int_{A} \{n \cdot (\beta_x (x_{air} - x_{wall}))\} dA \tag{5}$$

where

β_x = surface moisture transfer coefficient (SMTC), $m \cdot s^{-1}$

x_{air}, x_{wall} = vapor fraction of the air in the center of the control volume and at the wall surface, respectively, $kg \cdot kg^{-1}$

A = surface, corresponding to the height and width of the control volume, m^2

An upwind scheme is applied for the discretization of the resulting system of equations. The airflow model is implemented in the CHAMPS-BES program (Nicolai and Grunewald 2006), which is an envelope model for the coupled simulation of heat, air, moisture, and pollutant transport in building components.

Notice that long-wave radiation among the surfaces in the room is considered in neither the CFD nor subzonal airflow models. The modeling of thermal radiation in CFD and subzonal models requires the implementation and application of two different radiation models. In general, CFD software incorporates standard models for thermal radiation, while other models are available for the implementation in subzonal models. The use of different radiation models may result in deviations between the CFD results and the subzonal model's results, which are caused by the radiation models. An analysis of the performance of different radiation models falls outside the scope of this paper.

Flow Element Subzones. If a subzone is under direct influence of a flow driver, for example a fan or a heater, the flow in the subzone is modeled as a flow element. Flow elements are treated as isolated volumes, where air movement is controlled by a restricted number of parameters and is fairly independent of the general flow in the enclosure. Often, the mathematical equations governing the airflow in flow elements are based on empirical relationships (Inard et al. 1996; Stewart and Ren 2006; Rajaratnam 1976). In this study,

a thermal boundary layer model is used, based on experimental work that focuses on the analysis of the thermal boundary layer along flat plates.

When a surface in a room is poorly insulated, the surface temperature is different from the surroundings and there is free convection between the surface and the surrounding air. In this case, the thickness of the boundary layer is zero at the top of the vertical surface and increases in the downward direction due to entrainment of room air (Figure 2). If the surface is located in calm surroundings, the boundary layer flow at the top of the surface will be laminar, and at a certain distance from the top it will become turbulent. The ratio between the buoyancy and viscous (friction) forces is expressed by the Grashof number (Gr) (Equation 6). Depending on whether the airflow in the boundary layer of a cold wall is laminar ($Gr_y < 1 \cdot 10^9$) or turbulent ($Gr_y > 1 \cdot 10^9$), Equations 7 and 9 or Equations 8 and 10 are applied for the boundary layer thickness and the flow through the boundary layer, respectively.

$$Gr_y(y) = \frac{g\beta\Delta T(H-y)^{-3}}{\nu^2} \tag{6}$$

$$\delta(y) = 0.048(H-y)^{1/4}\Delta T^{1/4} \tag{7}$$

$$\delta(y) = 0.11(H-y)^{7/10}\Delta T^{-1/10} \tag{8}$$

$$\phi(y) = 0.0024(H-y)^{3/4}\Delta T^{1/4}L \tag{9}$$

$$\phi(y) = 0.0021(H-y)^{6/5}\Delta T^{2/5}L \tag{10}$$

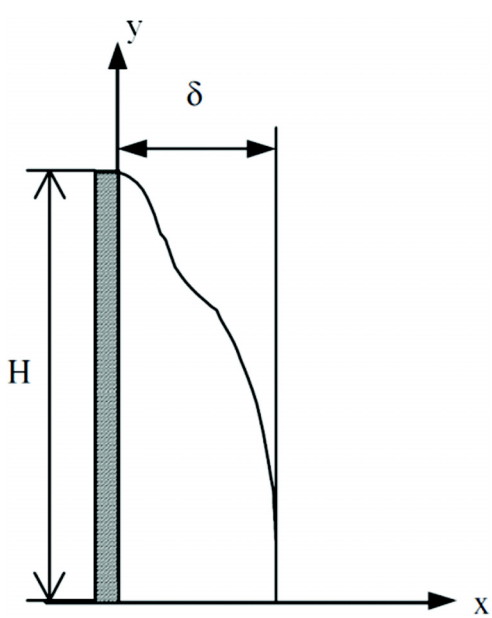

Figure 2 *Thermal boundary layer flow along a cold vertical surface (Stewart and Ren 2006).*

where

g = gravitational acceleration, $m \cdot s^{-2}$
β = thermal expansion coefficient of air, K^{-1}
ν = kinematic viscosity of air, $m^2 \cdot s^{-1}$
δ = boundary layer thickness, m
ϕ = volume flow rate through the boundary layer, $m^3 \cdot s^{-1}$
H = height of the wall, m
L = width of the wall, m
ΔT = temperature difference between the wall surface and the average temperature in the room, K
y = vertical coordinate, m

Local Convective Surface Heat Transfer Coefficients

Different models for the local convective surface heat and moisture transfer coefficients α_c and β_x, respectively, are applied. The relationships are determined analytically, experimentally, or numerically. Models are based on the relationships that resulted from a review of the literature on convective surface heat transfer coefficient modeling. The models are characterized by the different flow regimes in a room.

The local convective surface heat transfer (α_c) is then defined by the local Nusselt number (Nu) (Equation 11), which describes the ratio of convective to conductive heat transfer across the boundary.

$$\alpha_c = \frac{Nu_y \lambda}{y} \tag{11}$$

where

Nu_y = local Nusselt number along the building component,
λ = thermal conductivity of the fluid, $W \cdot m^{-1} \cdot K^{-1}$
y = coordinate along the component, m

Table 1 presents the convective surface heat transfer coefficient models that are applied, where Gr_y represents the local Grashof number, Pr is the Prandtl number for air, and Ra_y is the local Rayleigh number describing the ratio between conduction and convection. The table shows that three different models for the convective heat transfer coefficient (CHTC) are applied: a theoretical model (1) based on the boundary layer theory describing the natural convective airflow along a vertical flat plate with uniform surface temperature (Schlichting and Gersten 2003), and two experimentally based models (2) and (3). The models have been implemented, and the predicted CHTCs are compared with CFD predictions and average convective surface transfer coefficients obtained from literature (Beausoleil-Morrison 2000).

Local Convective Surface Moisture Transfer Coefficients

The moisture fluxes between the room and the building component are modeled using the Chilton-Colburn analogy

(Equation 12) (Chilton and Colburn 1934), which relates the heat and mass transfer coefficients directly:

$$\frac{\alpha_c}{\beta^X} = \rho c_p \left(\frac{\mathrm{Sc}}{\mathrm{Pr}}\right)^{2/3} = \rho c_p \left(\frac{\alpha}{D}\right)^{2/3} = \rho c_p \mathrm{Le}^{2/3} \tag{12}$$

where Le is the Lewis number, defined as the ratio of thermal diffusivity α ($m^2 \cdot s^{-1}$) to mass diffusivity D ($m^2 \cdot s^{-1}$), and Sc is the Schmidt number, defined as the ratio of momentum diffusivity (viscosity) and mass diffusivity.

Local convective surface heat transfer coefficients are obtained from the CHTC models, resulting in local convective surface moisture transfer coefficients (CHTCs and SMTCs).

RESULTS

The MINIBAT case is simulated using the presented subzonal airflow model with a model describing the flow in the boundary layer near the walls. The simulation results are compared with experimental results and results obtained from a CFD simulation.

Local Temperature and Relative Humidity

The results from the subzonal airflow model (Figure 5) are compared to the experimental (Figure 3) and CFD results (Figure 4). The results from the CFD simulation have been verified and validated based on work published by Inard et al. (1996). Results from the CFD simulations showed good agreement with the experimental results. As it is not within the scope of the current work to give an intensive validation of the CFD simulation, a detailed verification of the CFD results is omitted.

Comparing results from the subzonal model with the experimental and numerical results from CFD, the following observations are presented:

Global Distribution. The predicted temperature and vapor content distribution resulting from the measurements, CFD model, and subzonal model are relatively similar. The models are capable of giving a relatively rough prediction of the stratification in the room. However, it should also be noted that a qualitative comparison of the measured temperature and the resulting temperature from CFD and the subzonal model show clear deviations. A difference of approximately 1°C between the experimental and numerical data is observed. It is assumed that this systematic deviation is caused by the experimental accuracy of the investigations and by deviations of the numerical results. When this systematic deviation is neglected, CFD is best capable of predicting the temperature distribution in the room.

Near-Wall Distribution. While both models are capable of predicting a stratified pattern in the room, Figures 6 and 7 show that the subzonal model is capable of predicting within general a maximum relative deviation (δ_{max}) between 10%–15% for the temperature and vapor content near the walls (Equation 13).

$$\delta_{max} = \frac{T_{max} - T_{avg}}{T_{avg}} \tag{13}$$

where

T_{max} = maximum temperature along the wall, °C

T_{avg} = average temperature along the wall, °C

Convective Surface Transfer Coefficients

Besides local temperature and vapor content, convective surface transfer coefficients are important for the prediction of heat and moisture flows between room and walls. The subzonal model is used to model natural convective airflow in the room. The results obtained from the subzonal airflow model are used as input data for the surface transfer coefficient models. The predicted convective surface heat and moisture transfer coefficients along the walls resulting from the surface transfer coefficient models and the CFD model are compared. Table 2 presents an overview of the simulated surface transfer coefficient models and computational grids that are used.

Figures 8 and 9 present a comparison of the local convective surface heat and moisture transfer coefficients resulting from the different surface transfer coefficient models (Table 2) and the values obtained from the CFD simulation. With respect to the coefficients predicted by model (a) based on the flat plate analogy, the figures show an underprediction in the region from the leading corner down from/up to the center of the wall (y = 1.25 m) and an overprediction of the coefficients further from the center (y = 1.25). Comparison of the results with the CFD results showed that the main reason for the

Table 1. Local CHTC Models for Natural Convection

CHTC Model	Source		
(1) Flat plate (Schlichting and Gersten 2003)	Theory	Laminar ($\mathrm{Gr}_y < 1 \cdot 10^9$)	$\mathrm{Nu}_y = \frac{0.676\mathrm{Pr}^{1/2}}{(0.861+\mathrm{Pr})^{1/4}}\left(\frac{\mathrm{Gr}_y}{4}\right)^{1/4}$
		Turbulent ($\mathrm{Gr}_y > 1 \cdot 10^{10}$)	$\mathrm{Nu}_x = 0.0295(\mathrm{Gr}_y)^{2/5}(\mathrm{Pr})^{7/15}(1+0.494\mathrm{Pr}^{2/3})^{-2/5}$
(2) Turner and Flake (1980)	Experiment	$3.5 \cdot 10^6 < \mathrm{Ra} < 5.5 \cdot 10^9$	$\mathrm{Nu}_y = 0.524(\mathrm{Gr}_y)^{0.26}$
(3) Bohn et al. (1984)	Experiment	$3 \cdot 10^9 < \mathrm{Ra} < 6 \cdot 10^{10}$	$\mathrm{Nu}_y = 0.62(\mathrm{Ra}_y)^{1/4}$

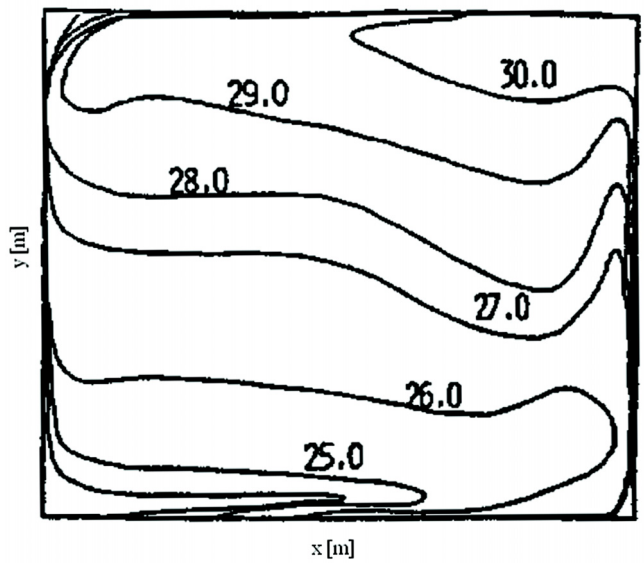

Figure 3 *Measured temperature distribution (°C) in the room (Inard et al. 1996).*

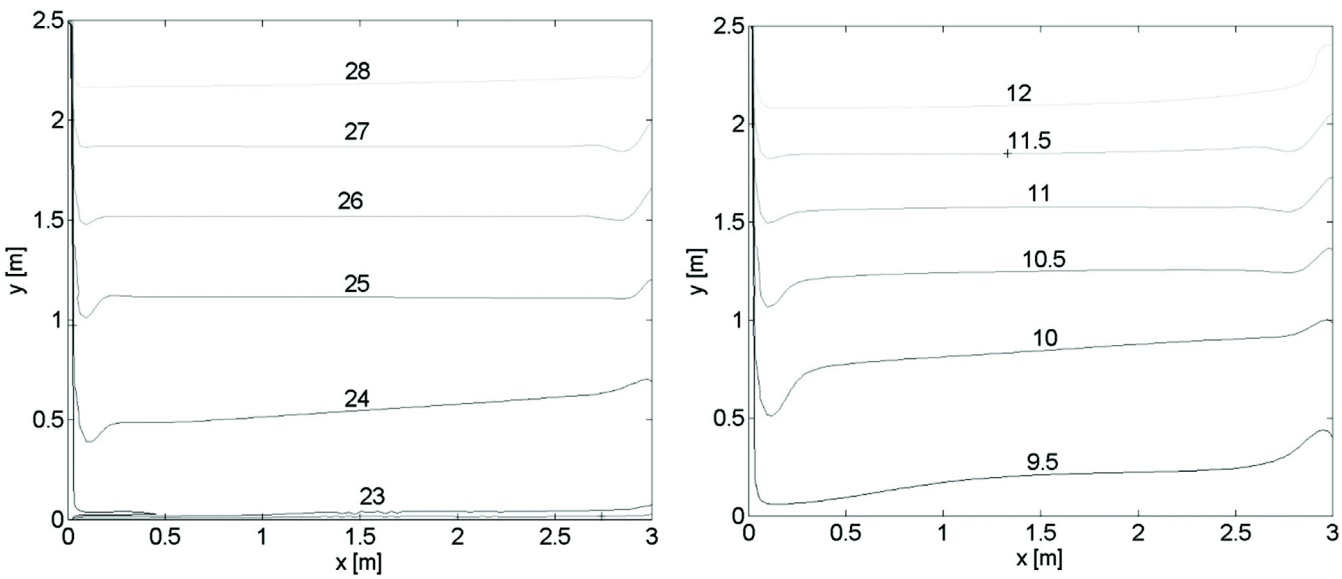

Figure 4 *The predicted temperature distribution (left) and vapor content distribution (mass fraction, x [g·kg^{-1}]) (right) in the room obtained from CFD.*

under/over prediction is that the size of the laminar region is overpredicted by the model, resulting in smaller surface transfer coefficients, while the size of the turbulent region is underpredicted.

Regarding the results predicted by models (b), (c), and (d), the resulting local surface transfer coefficients are comparable with the coefficients predicted by CFD. The surface transfer coefficients predicted by models (b) and (c), based on Turner and Flake (1980), give the best agreement, while the deviation is less than 10%. Model (d) (Bohn et al. 1984) gives a slight overprediction and a maximum relative deviation of approximately 25%. This relatively high deviation might be caused by the dissimilarity between the simulated case and the conditions that have been used for the determination of the relationships. Bohn et al. (1984) determined the CHTC for a cube in water, with a rib length of 0.3 m and a range of

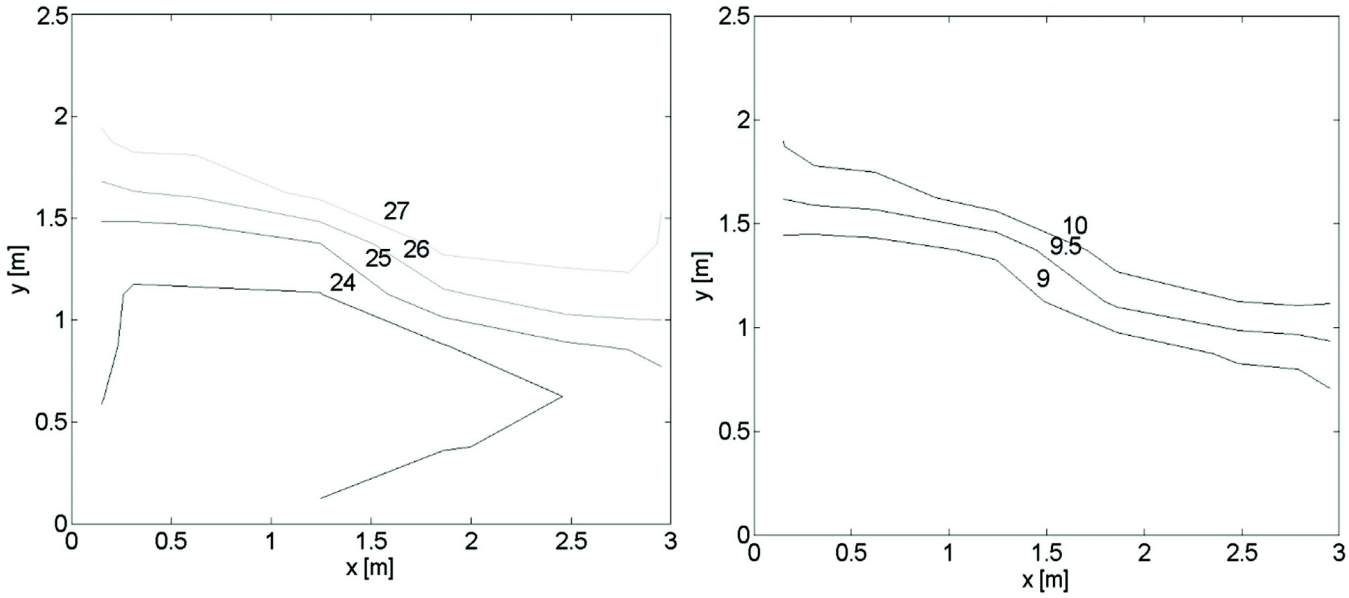

Figure 5 *Temperature distribution (°C) (left) and vapor content distribution ($g \cdot kg^{-1}$) (right) predicted by the subzonal model (b).*

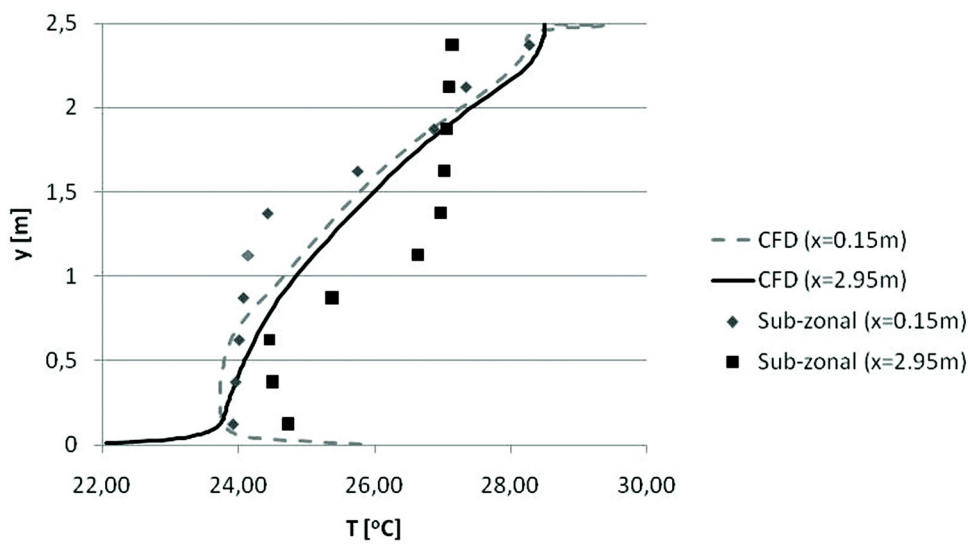

Figure 6 *Temperature (°C) distribution at different locations in the room (x = 0.15 and 2.95 m).*

Rayleigh number between $3 \cdot 10^9$ and $6 \cdot 10^{10}$, while Turner et al. (1980) determined the CHTCs for various rectangular boxes in air, and a range of Rayleigh number between $3.5 \cdot 10^6$ and $5.5 \cdot 10^9$. The Rayleigh number in the studied room varied between $2.5 \cdot 10^6$ and $18 \cdot 10^9$.

The investigations showed that the surface transfer coefficient model based on the flat plate analogy is not suitable for prediction of the convective surface transfer coefficients in the room. As was discussed earlier, the specific assumptions of the boundary layer theory for flat plates, for example regarding the boundary conditions, geometrical influences, entrance velocity and leading edges, and surface roughness, are not (entirely) valid in building enclosures. Similar observations were reported by Khalifa (2001), and the authors concluded

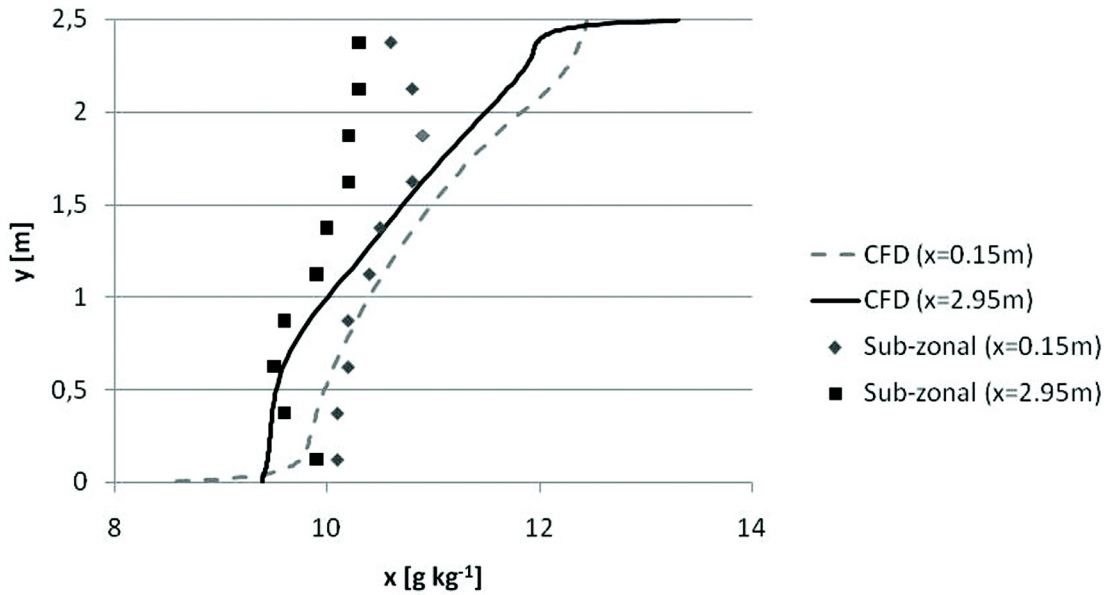

Figure 7 *Vapor content ($g \cdot kg^{-1}$) distribution at different locations in the room (x = 0.15 and 2.95 m).*

Table 2. Surface Transfer Coefficient Models

Model		Grid (*x* × *y*)
(ref)	Beausoleil-Morrison (2000)	8 × 10
(a)	Flat plate	8 × 10
(b)	Turner and Flake (1980)	8 × 10
(c)	Turner and Flake (1980)	16 × 20
(d)	Bohn et al. (1984)	8 × 10

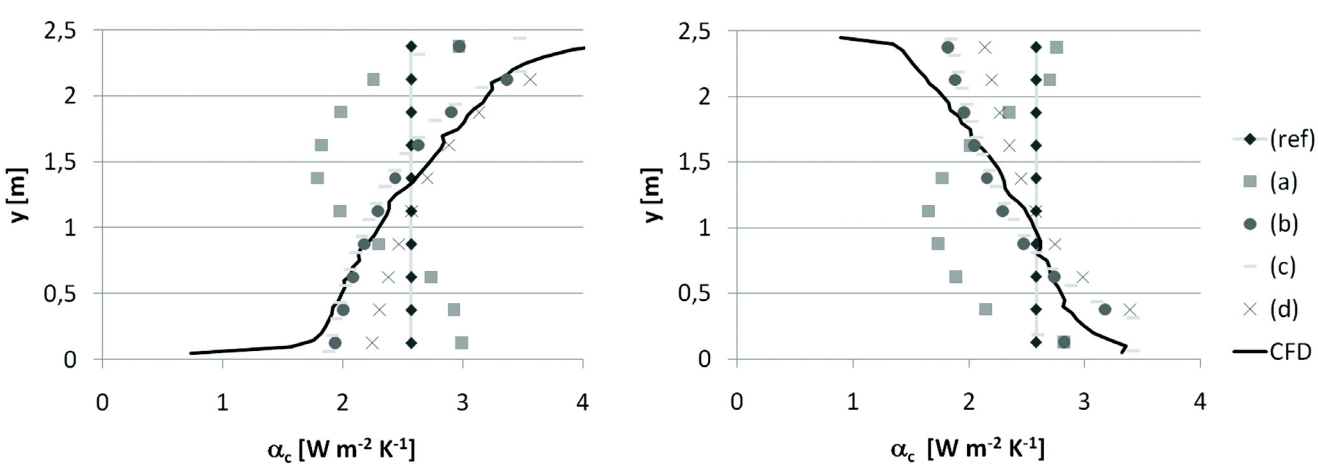

Figure 8 *Convective surface heat transfer coefficient (c) ($W \cdot m^{-2} \cdot K^{-1}$) for the western wall (left) and the eastern wall (right).*

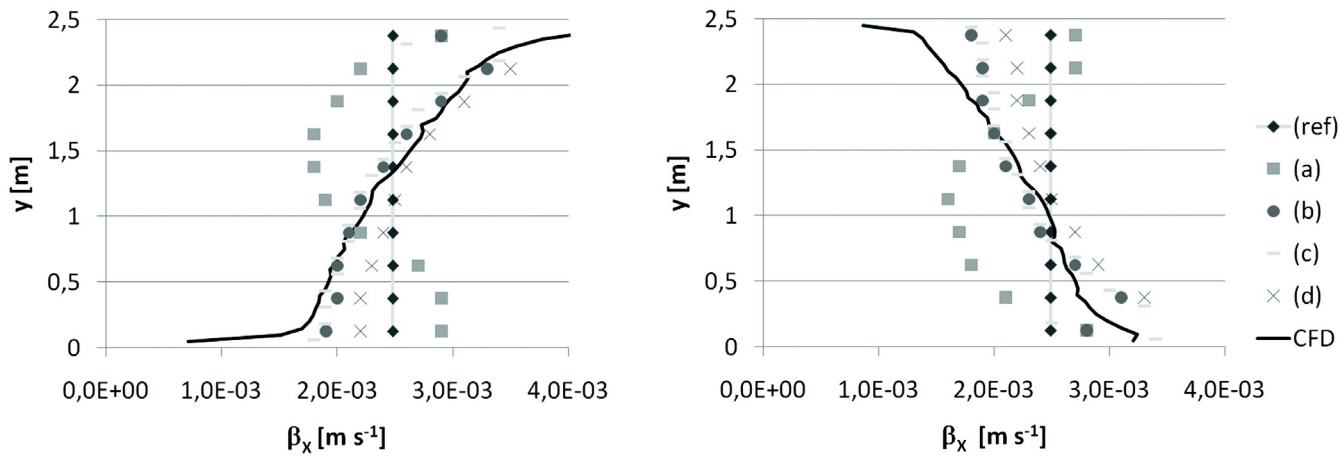

Figure 9 *Convective surface moisture transfer coefficient for the western wall (left) and the eastern wall (right).*

that a correlation obtained for an isolated flat plate is not suitable for a surface in a real-sized enclosure, especially for buildings. This conclusion was confirmed by the present investigations. Furthermore, the boundary layer model that has been developed based on measurements of the global indoor environmental conditions for natural convection in a room, such as the model developed by Turner et al. (1980), is suitable for prediction of the convective surface transfer coefficients, provided similar Rayleigh numbers are observed as in the experimental conditions on which the correlation is based.

CONCLUSIONS AND DISCUSSION

This paper presents results of investigation of the applicability of the subzonal model for transient HAM building simulations. A case study considering the natural convective airflow in a room was selected for analysis. Subzonal airflow models and surface transfer coefficients models were simulated in order to predict the local indoor environmental conditions and convective surface transfer coefficients in the room. The numerical results obtained from these models were compared with both experimental and numerical results obtained from a CFD.

The investigation showed that the subzonal airflow model is able to predict the natural convective airflow in a room. The application of a thermal boundary layer model was obvious and resulted in a relatively accurate prediction of the local indoor temperature and relative humidity with a maximum relative deviation of 10%–15% (Equation 13). Other researchers (Wurtz et al. 1999; Mora et al. 2003) also observed that the subzonal airflow model gave an accurate prediction of the global temperature distribution for natural convection in a room.

With respect to the prediction of the local convective surface transfer coefficients, the model based on the experimental correlations for natural convection in an enclosure developed by Turner et al. (1980) gave a prediction with a maximum relative deviation up to 10%.

In conclusion, subzonal models combined with an appropriate surface transfer coefficient model are able to give a prediction of the indoor environmental conditions in a room under natural convective conditions. However, one important remark should be made. In the case studies, reference conditions, for example experimental data or numerical results from CFD, were used for the development of a reliable subzonal airflow model. The availability of such reference conditions is a prerequisite for the development of reliable subzonal and surface transfer coefficient models. Additional information for forced and mixed convection in a room can be found in Steskens (2009).

The main advantage of the subzonal model is a significant reduction in computational effort compared to CFD. The computation time of a subzonal airflow model with a surface transfer coefficient model implemented generally varies from a few seconds up to 20 seconds for one time step. The subzonal airflow model is solved on a relatively coarse grid, while only three equations describing the conservation of mass, energy, and vapor are solved per time step. The computational effort of the CFD simulations that have been carried out is relatively large. The computation time of a CFD simulation varies from several hours up to a few days. Furthermore, the stability of the subzonal model showed to be relatively large compared to CFD, resulting in only a few iterations for solving the airflow and the temperature and vapor content fields. The relatively short computation time and flexibility makes the application of the subzonal model attractive for the transient simulation of heat, air, and moisture in buildings.

REFERENCES

Allard, F., J. Brau, C. Inard, and J.M. Pallier. 1987. Thermal experiments of full-scale dwelling cells in artificial climatic conditions. *Energy and Buildings* 10:49–58.

Beausoleil-Morrison, I. 2000. The adaptive coupling of heat and air flow modeling within dynamic whole-building simulation. PhD thesis, University of Strathclyde, Glasgow, UK.

Bohn, M.S., A.T. Kirkpatrik, and D.A. Olson. 1984. Experimental study of three-dimensional natural convection and high Rayleigh numbers. *ASME Trans J Heat Transfer* 106:339–84.

Chilton, T.H., and A.P. Colburn. 1934. Mass transfer (absorption) coefficients. *Industrial and Engineering Chemistry* 26:1183–7.

Inard, C., H. Bouia, and P. Dalicieux. 1996. Prediction of air temperature distribution in buildings with a zonal model. *Energy Build*. 24:125–32.

Khalifa, A.N. 2001. Natural convective heat transfer coefficient—A review: Part I, Isolated vertical and horizontal surfaces. *Energy Conversion and Management* 42:491–504.

Megri, A.C., and F. Haghighat. 2007. Zonal modeling for simulating indoor environment of buildings: Review, recent developments, and applications. *HVAC&R Research* 13:887.

Mendonça, K.C. 2004. Modelisation thermo-hydro-aeraulique des locaux climatises selon l'approche zonale. PhD thesis, Université de La Rochelle, France.

Mora, L., J. Gadgil, and E. Wurtz. 2003. Comparing zonal and CFD model predictions of isothermal indoor airflows to experimental data. *Indoor Air* 13:77–85.

Mortensen, L.H. 2007. Hygrothermal microclimate on interior surfaces of the building envelope. PhD thesis, Technical University of Denmark (DTU).

Nicolai, A., and J. Grunewald. 2006. CHAMPS-BES Program for Coupled Heat, Air, Moisture and Pollutant Simulations in Building Envelope Systems. (User Manual). Building Energy and Environmental Systems Laboratory, Department of Mechanical and Aerospace Engineering, Syracuse University, New York.

Rajaratnam, N. 1976. *Turbulent Jets*. New York: Elsevier Scientific Publishing Company.

Schlichting, H., and K. Gersten. 2003. *Boundary Layer Theory*, 8th Edition. New York: Springer.

Steeman, H.J. 2008. Modelling local hygrothermal interaction between airflow and porous materials for building applications. PhD thesis, Universiteit Gent, Belgium.

Steskens, P.W.M.H. 2009. Modelling of the hygrothermal interactions between the indoor environment and the building envelope. PhD thesis, Department of Civil Engineering (BYG), Technical University of Denmark (DTU).

Stewart, J., and Ren, Z. 2006. COwZ-A subzonal indoor airflow, temperature and contaminant dispersion model. *Build. Environ*. 41:1631–48.

Teshome, E.J., and F. Haghighat. 2004. Zonal models for indoor air flow—A critical review. *International Journal of Ventilation* 3:119.

Turner, B.L., and L.D. Flake. 1980. The experimental measurement of natural convection heat transfer in rectangular enclosures. *ASME Trans, J Heat Transfer* 102:236–41.

Woloszyn, M., and C. Rode. 2008. Tools for performance simulation of heat, air and moisture conditions of whole buildings. *Building Simulation* 1:5–24.

Wurtz, E., L. Mora, and C. Inard. 2006. An equation-based simulation environment to investigate fast building simulation. *Building and Environment* 41:1571–83.

Wurtz, E., J. Nataf, and F. Winkelmann. 1999. Two- and three-dimensional natural and mixed convection simulation using modular zonal models in buildings. *Int. J. Heat Mass Transfer* 42:923–40.