



Aalborg Universitet

AALBORG UNIVERSITY
DENMARK

CLIMA 2016 - proceedings of the 12th REHVA World Congress

volume 5

Heiselberg, Per Kvols

Publication date:
2016

Document Version
Publisher's PDF, also known as Version of record

[Link to publication from Aalborg University](#)

Citation for published version (APA):

Heiselberg, P. K. (Ed.) (2016). *CLIMA 2016 - proceedings of the 12th REHVA World Congress: volume 5*. Department of Civil Engineering, Aalborg University.

General rights

Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

- Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
- You may not further distribute the material or use it for any profit-making activity or commercial gain
- You may freely distribute the URL identifying the publication in the public portal -

Take down policy

If you believe that this document breaches copyright please contact us at vbn@aub.aau.dk providing details, and we will remove access to the work immediately and investigate your claim.

The Impact of Thermal Radiation on Condensation on Cold Surfaces of Ventilated Rooms

Catalin Teodosiu¹, Viorel Ilie², Raluca Teodosiu³

*Thermo-Hydraulic and Protection of the Atmosphere Systems Department,
Technical University of Civil Engineering
Lacul Tei Blvd., no. 122-124 Bucharest, 020396, Romania*

¹cteodosiu@yahoo.com

²viorel_ilie_88@yahoo.com

³ralucahohota@yahoo.com

Abstract

The objective of this study is to investigate the impact of radiative heat transfer on the global exchange of thermal energy between indoor surfaces and air in rooms. In addition, this paper is focusing on the influence of this phenomenon on water vapor condensation on cold surfaces. The approach is based on CFD (Computational Fluid Dynamics) technique, using an integrated moisture–heat–air flow methodology, required for comprehensive thermal investigations and condensation mechanisms. Basically, transport equations for radiation intensity to solve the radiative heat transfer and equations expressing the conservation of the water vapor mass fraction are added to the equations governing a turbulent confined non-isothermal airflow. The numerical model is applied in the case of a room with mixing ventilation and radiant floor heating system. One of the vertical walls of the enclosure is assumed to be cold to favor the condensation on its surface. For the purpose of this study, this configuration has been investigated with and without considering the radiation model. The results are focusing on comparing the apparition of the condensate, its amount and its distribution on the cold wall, with and without considering the radiative part of heat transfer in the room. It can be concluded that it is not safe to disregard the radiative part of heat transfer in indoor air simulations when condensation risk investigations are required.

Keywords – CFD (Computational Fluid Dynamics) modeling; condensation risk; thermal radiation

1. Introduction

The proper design of buildings services can be achieved only by taken into account three main criteria: thermal comfort of occupants, indoor air quality, and energy efficiency. The indoor air humidity has a direct effect on all these issues. For example, people may feel unpleasant sensation of dryness for relative humidity less than about 30% [1,2]. On the other hand, high relative humidity (above 70%), associated with high air temperatures, leads to a feeling of stuffiness, while high relative humidity, combined with

low air temperatures, intensifies the sensation of cold [3]. High moisture content of indoor air, correlated with cold weather, is also causing condensation on cold surfaces or even within the materials. Consequently, high levels of relative humidity in rooms (greater than 75%), with favorable conditions of temperature, contribute to mold development [4]. This is a major human health concern, clearly confirmed nowadays [5]. In addition, condensation in building materials may result in premature deterioration of these materials [6]. On the contrary, very dry indoor air leads to frequent electrostatic shocks [7]. Finally, the level of humidity in rooms directly influences the latent load of HVAC (heating, ventilation and air conditioning) systems when controlling indoor relative humidity.

Despite all this, the number of studies dealing with phenomena related to indoor air humidity is quite low. In addition, the investigations on humidity aspects including condensation in buildings are generally lacking. Given that condensation mechanisms occur locally and these phenomena deeply depend on local psychrometric conditions (temperature, humidity) but also on local air flows, detailed moisture-heat-airflow models are required. In this context, the CFD (Computational Fluid Dynamics) technique is the appropriate solution as this approach allows detailed predictions of temperature and velocity fields in rooms. This is possible due to significant development and improvement of CFD simulations in the last 30 years that have made the CFD technique an usually tool for forecasting air flow in enclosures [8]. Consequently, the CFD models are increasingly used nowadays to perform detailed heat, air and moisture transfer analyses in indoor environments [9-12]. As a result, the aim of this study is to propose a model that takes into account wall surface condensation phenomena in CFD simulations for buildings. On the other hand, the accurate calculation of surface temperature is essential for the correct prediction of surface condensation and its development. Consequently, the approach to assess the surface temperature should take compulsory into account the sensible heat transfer (including conduction, convection and radiation) and latent heat transfer of that surface. As radiation heat transfer effects are usually neglected in studies on air movement in rooms, this study also investigates if the radiative part of heat transfer in indoor air CFD simulations could not be considered when condensation risk investigations are performed.

This analysis is carried out in this study in the case of a small room equipped with a mixing ventilation and radiant floor heating system.

2. Numerical Model

2.1. Airflow and surface condensation model

The integrated moisture-energy-air flow approach is carried out through a CFD model adding an equation expressing the conservation of the water vapor mass fraction to the basic equations governing a turbulent confined

non-isothermal flow of humid air (considered as a mixture dry air-water vapor). This equation can be written in a similar manner to classical convection-diffusion CFD equations:

$$\rho \frac{\partial}{\partial x_i} (u_i m_i) + \frac{\partial}{\partial x_i} J_{i,i} = S_i \quad (1)$$

where the left-hand side terms stand for the convective term (ρ - density of the humid air, x_i - spatial coordinate, u_i - velocity component in i direction, m_i - water vapor mass fraction) and diffusion term respectively ($J_{i,i}$ - water vapor diffusion flux), while the right-hand side term S_i represents potentially water vapour source terms.

The diffusion term in (1) takes into account both molecular diffusion and turbulent diffusion mechanisms:

$$\frac{\partial}{\partial x_i} J_{i,i} = \rho \frac{\partial}{\partial x_i} \left(D_{i,m} \frac{\partial m_i}{\partial x_i} \right) - \frac{\partial}{\partial x_i} \left(\overline{-u'_i m'_i} \right) \quad (2)$$

where $D_{i,m}$ is the water vapor molecular diffusion coefficient and $u'_i m'_i$ is the turbulent mass flux of water vapor, u'_i being the velocity fluctuation.

The molecular diffusion in (2) is represented by Fick's first law (diffusion flux is proportional to the concentration gradient). The value of water vapor diffusion coefficient in air is considered constant in the model ($2.55 \times 10^{-5} \text{ m}^2/\text{s}$).

The turbulent diffusion in (2) is taking into account through the turbulence model used to describe the flow of the moist air in the CFD model. Consequently, the turbulent mass flux of water vapor is predicted using a methodology similar to that of the Reynolds analogy: the turbulent mass diffusivity (D_t) is associated with the turbulent viscosity (μ_t) using the turbulent Schmidt number (Sc_t) – see (3) and (4).

$$-\overline{u'_i m'_i} = \frac{\mu_t}{Sc_t} \frac{\partial m_i}{\partial x} \quad (3)$$

$$Sc_t = \frac{\mu_t}{\rho D_t} \quad (4)$$

In addition, the convection and diffusion phenomena related to humidity are studied in the CFD model using the following hypotheses: fluid taking into account (humid air): incompressible Newtonian fluid; humid air: mixture of two perfect gases (dry air and water vapor); there is no chemical reaction between the constituents of the mixture; heat and mass transfer mechanisms in the mixture are negligible; mixture density, ideal gas law formulation (based on the mixture temperature and the concentration of each component in the mixture); mixture specific heat capacity: mixing law formulation, based on mass fraction average of the

two species (air and water vapor) heat capacities; mixture thermal conductivity and viscosity: determined by means of kinetic theory.

Concerning the surface condensation, the model is based on the approach suggested by International Energy Agency [13]. The condensed vapor density flux is computed based on the assumption that the water vapor transport in air is mainly due to convective mechanisms:

$$\Phi_{vap.cond} = \beta(P_{vap.air} - P_{vap.surface}) \quad (5)$$

where β (s/m) - proportionality coefficient that defines the water vapor diffusion between the indoor air and the walls surface, $p_{vap.air}$ - vapor pressure in air, and $p_{vap.surface}$ - vapor pressure on the wall surface.

The coefficient β from (2) is correlated with the convective heat transfer coefficient for applications in the field of thermal building [13]. Consequently, this coefficient depends on the convective heat transfer coefficient, resulting from heat exchanges computation between air and walls in CFD simulations. The convective heat transfer coefficient is computed in the model taking into account the faces (on the walls) of the discretization mesh and the centers of the first discretization elements within the indoor air (viscous zone). As a result, for the viscous sublayer (e.g. cell with area S_F and temperature T_F), the convective heat exchange takes the form:

$$\Phi = h_c(T_w - T_{air})S = h_c(T_F - T_{C0})S_F \quad (6)$$

where T_w – wall surface temperature, T_{air} – air temperature, and S – surface; T_{C0} – temperature in the barycenter of the first discretization element (near the wall).

On the other hand, the viscous region near the ceiling is characterised in terms of heat exchanges by conduction, the convection playing a negligible role. Consequently, the heat flux density can be taken into account by the classical Fourier law, which means that the convective heat transfer coefficient can be determined as follows:

$$\varphi = -\lambda_{air} \overrightarrow{grad}T \vec{n} = \lambda_{air} \left(\frac{\partial T}{\partial n} \right) = h_c(T_F - T_{C0}) \Rightarrow h_c = \frac{\lambda_{air}}{(T_F - T_{C0})} \left(\frac{\partial T}{\partial n} \right) \quad (7)$$

where λ_{air} - thermal conductivity of the humid air.

Temperature gradient in (7) is calculated based on the following approach:

$$\left(\frac{\partial T}{\partial n} \right) = \frac{(T_F - T_{C0})}{S_F} \alpha \quad (8)$$

where S_F represents the area of the discretization element on the wall and α is computed according to (9).

$$\alpha = \frac{\overrightarrow{S_F} * \overrightarrow{S_F}}{\overrightarrow{S_F} * \overrightarrow{S_0}} \quad (9)$$

with $s_0 = (\text{coord}_F - \text{coord}_C)$

Based on (7), the value of the convective heat transfer coefficient is determined, for a discretization cell that has one face on the wall, by taking into account the following data: temperature difference between the center of the surface discretization element (on the wall) and the centroid of the cell which contains this surface discretization element; distance between the center of the surface discretization element and the centroid of the cell enclosing this surface discretization element; the surface of the discretization element; thermal conductivity of the moist air located in the volume finite centroid [14].

The mass flow rate of water vapor condensed on the ceiling $\dot{m}_{vap.cond}$ is calculated, for each discretization element that is positioned on the wall, once the coefficient β is solved (based on its correlation with the convective heat transfer coefficient – see above), as follows [14]:

$$\dot{m}_{vap.cond} = \frac{dm_{liq.surface}}{dt} \quad (10)$$

if

$$P_{vap} - P_{vap.sat} > 0$$

$$\dot{m}_{liq.surface} = 7,4 \times 10^{-9} h_c S_{F_i} (P_{vap} - P_{vap.sat})$$

else

$$\dot{m}_{liq.surface} = 0$$

where $m_{liq.surface}$ - condensed vapor flux, based on (5), $p_{vap.sat}$ - saturation pressure of water vapour, and p_{vap} - partial pressure of water vapor.

Finally, using the mass flow rate of water vapor condensed on the surface, the link with the volume computation should be completed by taking into account the mass balance and the energy balance. The mass balance is carried out in the following way: the condensate flow rate is removed by means of source term that is introduced in (1), the water vapor conservation equation of the CFD model, On the other hand, the energy balance is based on sink terms added in the energy conservation equation of the CFD model. These terms are related to latent heat of vaporization and sensible heat of the removed condensed water vapor [14].

2.2. Radiation model

The radiative heat transfer modeling is based on the assumption that the indoor air is a non-participating radiation medium despite of the fact that humid air contains water vapor which is an absorbing element in the infrared. This hypothesis has been used with satisfactory results in other studies [15]. In addition, all surfaces taken into account within the model

have the following properties: grey scattering surfaces (transmission and reflection).

In order to assess the radiative heat fluxes (multiple reflections that take place between the walls), the discrete ordinates (DO) radiation model in its conservative version (finite-volume method) [16] is employed. This approach solves the radiative transfer equation written as a transport equation for radiation intensity in the spatial coordinates:

$$\frac{dL(r_i, \vec{u})}{dx} + \frac{dL(r_j, \vec{u})}{dy} + \frac{dL(r_k, \vec{u})}{dz} = \kappa(r_i)[L^0[T(r_i)] - L(r_i, \vec{u})] \quad (11)$$

where L – radiation intensity, $r(r_i, r_j, r_k)$ – position vector, u – direction vector, κ – absorption coefficient, T – absolute temperature and L^0 – black body radiation intensity given by:

$$L^0[T(r_i)] = \sigma \frac{T^4(r_i)}{\pi} \quad (12)$$

where σ – Stefan-Boltzmann constant ($5.67 \times 10^{-8} \text{ W/m}^2\text{K}^4$).

Equation (11) is added to the principal equations governing the humid air flow in the CFD model. Furthermore, the radiation intensity transport equation (11) is numerically solved as the humid air flow CFD model equations.

The DO radiation model resolves (11) for a finite number of discrete solid angles (the number of equations solved is given by the amount of discrete solid angles defined). The solid angles are obtained based on an angular discretization, using control angles that determine the discretization of each octant in the angular space.

It must be mentioned that the same methodology has been successfully used in other similar studies [17,18].

2.3. Numerical model summary

The airflow model, surface condensation model, and radiation model have been developed using the general-purpose, finite-volume, Navier-Stokes solver Ansys Fluent (version 15.0.0). The main features of the CFD model are presented in Table 1.

3. Case Study

The numerical model presented above is applied in the case of a small room with mixing ventilation and radiant heating system over the entire floor surface (Fig. 1).

The simulations have been performed for the air supply conditions given in Table 2. The thermal boundary conditions for the walls of the room (excluding the floor) are imposed based on a convective heat transfer, using

an external convective heat transfer coefficient and a supposed external temperature (Table 3).

Table 1. Numerical model overview

Characteristic	Description
Fluid	Humid air (mixture: dry air-water vapour)
Flow	Three-dimensional, unsteady, non-isothermal, turbulent
Computational domain discretization	Finite volumes, unstructured mesh (tetrahedral elements)
Turbulence model	Shear Stress Transport (SST) turbulent kinetic energy-specific turbulent dissipation rate ($k-\omega$), with low-Reynolds corrections
Radiation model	Discrete ordinates (DO), two control angles for the polar and azimuthal angles to locate the vector direction in space
Numerical solution	Second-order upwind scheme; Velocity-pressure coupling: SIMPLE algorithm; Convergence acceleration: algebraic multigrid

As shown in Table 3, the air supply opposite wall is assumed to be cold in order to favor the formation of condensation on its surface.

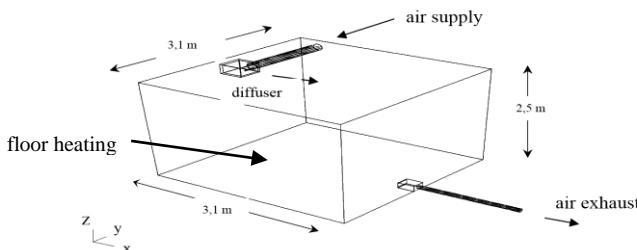


Fig. 1 Configuration of the case study

Table 2. Ventilation inlet characteristics

Air changes per hour (h^{-1})	Moisture content (g/kg)	Temperature (°C)
3.0	12.0	20.0

On the other hand, the surface of the floor is held at a fixed temperature, 29°C (Dirichlet boundary condition), in order to represent the functioning of floor heating.

For the purpose of this study, the simulations have been performed, for the configuration taken into account, with and without considering the radiative heat flux to the walls of the room.

Table 3. Thermal boundary conditions

Wall*	Convective heat transfer (W/m ² °C)	Temperature (°C)
South	12.0	20.0
North	20.0	10.0
East	12.0	20.0
West	12.0	20.0
Ceiling	12.0	20.0

* The wall behind the air supply is considered as the “south” wall of the room.

4. Results

The results are focusing on comparing the apparition of the condensate, its amount and its distribution in time on the cold wall, with and without taking into account the radiative part of heat transfer in the room (Figs. 2 and 3, respectively).

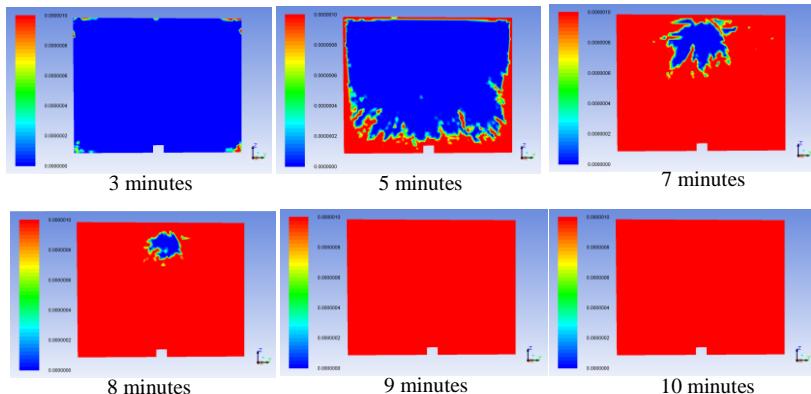


Fig. 2 Distribution in time of the condensate (I) on the cold wall
(without taking into account the radiative heat flux to the walls of the room)

As it can be seen, the differences between the two situations are important. In the case without thermal radiation, condensation occurs much faster (the condensate is clearly visible after 3 minutes), while for the simulations with thermal radiation flux between the walls of the room, roughly the same condensation is noticed after 7 minutes.

Furthermore, without considering the radiation, condensation occurs almost over the whole surface of the cold wall after only 8 minutes, while at

the same moment of time, considering the radiation leads to condensation only on the sides of the cold wall.

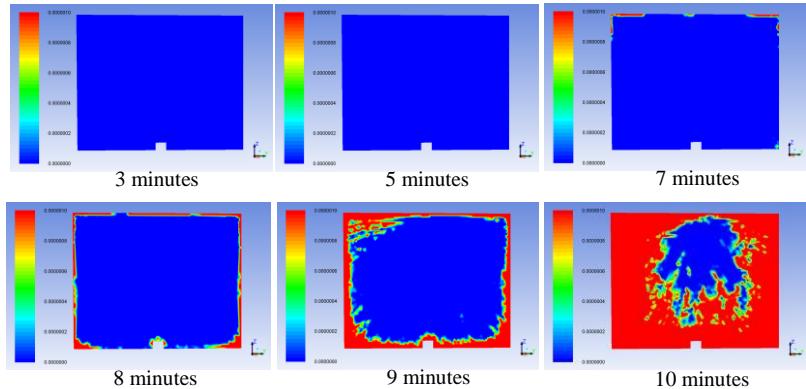


Fig. 3 Distribution in time of the condensate (I) on the cold wall
(taking into account the radiative heat flux to the walls of the room)

This is explained by a higher wall surface temperature throughout the simulation time when considering radiation (average temperature of about 13°C compared to 12°C without taking into account the radiative heat flux to the walls of the room), correlated with the same evolution in time of the room air dew point temperature, with and without radiation (average values from 10°C after 3 minutes, to around 12.5°C after 10 minutes, due to more humid air supplied in the room).

Finally, it can be observed in Figs. 2 and 3 that condensation does not take place at the beginning in the area where the air jet impinges the cold wall.

5. Conclusions

The numerical model developed in this study is capable to deal with moisture-heat-air flow phenomena occurring in ventilated rooms with radiant heating systems. In addition, the numerical model is useful in analyses concerning condensation of humid air on cold surfaces (distribution of condensate in time and space).

On the other hand, based on the results achieved, it can be concluded that it is not safe to neglect the radiative heat transfer in indoor air simulations when condensation risk investigations are required.

Acknowledgment

This work was supported by a grant of the Romanian Ministry of Education and Research, CNCS-UEFISCDI, project number PN-II-ID-JRP-RO-FR-2012-0071.

References

- [1] T. Alsmo and C. Alsmo. Ventilation and Relative Humidity in Swedish Buildings. *Journal of Environmental Protection*. 5 (2014) 1022–1036.
- [2] P. Wolkoff and S.K. Kjærgaard. The dichotomy of relative humidity on indoor air quality. *Environmental International*. 33 (2007) 850–857.
- [3] M. Woloszyn. Hygro-thermo-aeraulic modeling of multi-zone buildings. Proposal of a strategy for the resolution of the coupled system (in French). Ph. D. thesis, INSA Lyon, Lyon, France, 1999.
- [4] P. Johansson, I. Samuelson, A. Ekstrand-Tobin, K. Mjörnell, P.I. Sandberg and E. Sikander. Microbiological growth on building materials – critical moisture levels. State of the art. SP Report 2005:11, SP Swedish National Testing and Research Institute, Borås, Sweden, 2005. [Online: accessed 15 January 2015, Available: <http://www.kuleuven.ac.be/bwf/projects/annex41/protected/data/SP%20Oct%202005%20Prese%20A41-T4-S-05-7.pdf>.]
- [5] B. Weinhold. A Spreading Concern: Inhalational Health Effects of Mold. *Environmental Health Perspectives*. 115 (2007) A300–A305.
- [6] U. Haverinen. Modeling moisture damage observations and their association with health symptoms. Ph. D. thesis, National Public Health Institute, Department of Environmental Health, University of Kuopio, Finland, 2002.
- [7] R. Hohota. Moisture modeling in a CFD code (low velocity in large cavity) – comparison with the experimental (in French). Ph. D. thesis, INSA Lyon, Lyon, France, 2003.
- [8] D. N. Sørensen and P. V. Nielsen. Quality control of computational fluid dynamics in indoor environments. *Indoor Air*. 13 (2003) 2-17.
- [9] C. Teodosiu, R. Hohota, G. Rusaouën and M. Woloszyn. Numerical prediction of indoor air humidity and its effect on indoor environment. *Building and Environment*. 38 (2003) 655-664.
- [10] L. H. Mortensen, M. Woloszyn, C. Rode and R. Peuhkuri. Investigation of Microclimate by CFD Modeling of Moisture Interactions between Air and Constructions. *Journal of Building Physics*. 30 (2007) 279-315.
- [11] M. Van Belleghem, H. J. Steeman, M. Steeman, A. Janssens and M. De Paepe. Sensitivity analysis of CFD coupled non-isothermal heat and moisture. *Building and Environment*. 45 (2010) 2485-2496.
- [12] H. Huang, S. Kato, R. Hu and Y. Ishida. Development of new indices to assess the contribution of moisture sources to indoor humidity and application to optimization design: Proposal of CRI_(H) and a transient simulation for the prediction of indoor humidity. *Building and Environment*. 46 (2011) 1817-1826.
- [13] International Energy Agency. Condensation and Energy. Energy conservation in buildings and community systems programme. Report Annex 14, vol. 1, International Energy Agency (IEA), Paris, France, 1991.
- [14] R. Teodosiu. Integrated moisture (including condensation) – energy – airflow model within enclosures. Experimental validation. *Building and Environment*. 61 (2013) 197-209.
- [15] M. Tabarki and S. B. Mabrouk. The coupling in transient regime between the modelings of thermal and mass transfers inside a heated room and its radiator. *Heat Mass Transfer*. 48 (2012) 1889-1901.
- [16] R. Siegel and J.R. Howell. Thermal radiation heat transfer. Taylor & Francis, Washington DC, USA, 1992.
- [17] R. Gao, A. Li, O. Zhang and H. Zhang. Comparison of indoor air temperatures of different under-floor heating pipe layouts. *Energy Conversion and Management*. 52 (2011) 1295-1304.
- [18] A. Ploskic and S. Holmberg. Heat emission from thermal skirting boards. *Building and Environment*. 45 (2010) 1123-1133.