MEASUREMENT AND CFD ANALYSIS OF A LOCAL RADIANT COOLING SOLUTION

Shiva Najaf Khosravi*, Helene Teufl, Ardeshir Mahdavi

TU Wien, Department of Building Physics and Building Ecology, Karlsplatz 13, 1040 Vienna, Austria

* corresponding author: `shiva.nkhosravi@gmail.com`

Abstract. This paper entails an empirical and computational assessment of the air flow field in the close proximity of a vertically positioned radiant cooling panel. This radiant cooling solution differs from the conventional large-area radiant cooling systems (e.g., ceiling panels). It involves rather small-sized vertical panels positioned close to occupants. Moreover, the panels are designed so as to manage potential surface condensation of water vapor via integrated drainage elements. Hence, the panels can be operated with relatively low surface temperatures. The low panel surface temperature and its proximity to the occupants are intended to compensate for the potential lower cooling power due to the relatively small panel size. In this paper, we specifically explore the air flow field close to the local radiant cooling panel via laboratory measurements and CFD (Computational Fluid Dynamics). Thus, possible issues regarding discomfort due to draft and turbulence risk close to the radiant panel can be examined. To this end, a prototypical local radiant cooling panel was installed in a mock-up office room of a laboratory. During the experiments, the air flow speed was measured and simulated at several heights (between 10 and 110 cm from the floor) and distances (ranging from 1 to 50 cm from the radiant panel). The results allow for the evaluation of the draft discomfort risk as well as the reliability of CFD in reproduction of the measurement results. A further step involved the numeric analysis of the effect of the human model on the air flow patter.

Keywords: Radiant cooling panel, thermal comfort, air flow, CFD simulation.

1. Introduction and background

It has been suggested that climate change due to greenhouse gas (GHG) emissions is one of the most significant challenges facing humanity [1]. Specifically, the global reliance on non-renewable energy sources leads to an increasing concentration of greenhouse gases in the atmosphere and thus contributes to climate change. In this respect, particular attention should be given to the household sector as a major consumer of energy [2]. The move from uniformly conditioned indoor environments to smaller, individually controlled zones, can be advantageous in view of thermal comfort and energy efficiency [3]. Limitations of traditional cooling systems, such as noise, draft, vertical thermal gradients, and high energy demand necessitate the investigation of alternative cooling systems. In this context, the application of water in chilled radiant panels can be of interest [4]. These panels can be broadly classified based on their installation location. The most common form is the ceiling-mounted cooling panel.

Radiant cooling ceiling systems were investigated, among others, in the laboratories in European countries in the early 1990s [5]. They have been shown to have the potential to provide adequate thermal conditions [6]. In some systems, the pipes are inserted in the surface layer of a wall, ceiling, or floor [7]. Lately, the integration of radiant cooling into furniture (e.g., in desks) has received attention [8]. Thereby, the chilled water could circulate through the table top

to remove the sensible heat, while latent heat is removed by an all-air cooling system [6]. As compared to conventional cooling systems, radiant cooling panels have the potential to offer smaller vertical temperature gradients, lower air movement velocity, and reduced local discomfort [9]. Whereas conventional air-conditioning systems are mainly based on convection, radiant panels utilize a combination of radiation and convection [10].

The thermal performance of the cooling panels has been investigated via both experimental and computational methods. The thermal comfort of five human subjects in a laboratory test room equipped with cooling ceilings was analysed by Nagano and Mochida [11]. In their experiment, the temperature difference between the ceiling and the room air was less than 5 K. According to their findings, the mean radiant temperature for a supine human body should be applied in the design of ceiling radiant cooling. Another study compared the radiant cooling systems with air-conditioning systems in terms of thermal comfort, energy consumption, and cost. The results suggested that the radiant ceiling panel system is capable of creating a smaller vertical variation of air temperature and a more comfortable environment [12].

Experimental studies are of course highly instructive, but also rather expensive and time-consuming. If carefully applied, CFD-based numeric simulation can provide alternative or at least complementary means of investigation. Computational fluid dynamics has

become an important tool in the prediction of thermal comfort in occupied spaces [13]. Compared to the experimental techniques, CFD simulations can provide detailed values regarding the flow distribution and concentration fields in the whole domain, rather than just targeted points for data collection [14–16]. Nevertheless, proper CFD application necessitates sufficient grid resolution, accurate choice of numerical models, estimation of numerical errors, and control over other numerical parameters for precise verification and validation [17–19]. Despite the dramatic progress in its application, CFD has not replaced experimental and theoretical analyses, but represents a highly useful complementary tool [20–22]. As such, experimental studies have been conducted to probe the precision of numerical simulation of the indoor thermal environment [23].

Myhren et al. [24] applied CFD simulations for two office rooms equipped with separate heating and ventilation systems, to investigate potential cold draught problems, the differences in vertical temperature gradients, and air speed levels. Another study compared the energy efficiency of the cooling panel with the all-air cooling system via CFD simulation and experimental method. Thereby, the cooling panel was found to be more energy efficient [25]. Predicting thermal comfort by applying virtual human manikins in the CFD simulation has been reported by numerous studies [13]. According to our observation, there are no relevant standards in CFD simulations for the sizes, shapes, and postures of human model, and their application mostly depends on the purposes of the research works [26–29]. Because of the excessive simulation time, the majority of CFD studies have applied simple (e.g., rectangular) shapes. In some cases, the nature of the study necessitated a complex human model [30].

Within the framework of a recently completed research study, the potential of vertical radiant panels in the proximity of users was investigated and related technological requirements regarding envelope tightness and ventilation systems as well as water vapor condensation risk were evaluated [27]. As mentioned before, the personal radiant cooling panels are designed so as to manage potential surface condensation of water vapor via specific elements. One option is to remove the condensed water via a drainage system. However, in our experience, a much simpler option would be sufficient in most case, namely the placement of an easy to clean metal container underneath the vertical cooling panel.

In the present contribution, we explore the potential of CFD simulations to analyse the airflow patterns and the velocity fields inside a test room with a vertical radiant panel positioned in the close proximity of a workstation. Simulated results were compared with previously obtained measurements. The objective of the numeric analysis was to capture the air flow patterns and local discomfort risk. The study included also the air velocity field around a simplified human model.

Figure 1. Test room equipped with a cooling panel.

2. Approach

Predicting the thermal performance of a cooling panel is not a trivial task. In general, this system represents a mixed convection heat-transfer mode, including human body buoyancy-driven thermal plumes, and radiation through the panel and wall surface. To this end, a prototypical local radiant cooling panel was installed in a mock-up office room of a laboratory (see Figure 1). To deal with potential surface condensation, a metal container was positioned below the local cooling elements.

During the laboratory experiments, the ambient air temperature and relative humidity in the room were kept at $30 \pm 0.5\,°C$ and $40 \pm 3\,\%$, respectively. The target panel surface temperature was $10\,°C$. For these indoor air and panel surface temperatures, the cooling power of the radiant element was estimated to be roughly $100\,W$ per square meter panel area [27]. The air flow speed was measured and simulated at several heights (10, 35, 60, 85, and $110\,cm$ above the floor) and distances (1, 1.5, 2, 3, 4, 5, 10, 15, 20, 30, 40, and $50\,cm$ from the radiant panel) (see Figure 2) [31].

In this study, the potential discomfort risk due to draft was explored and was not found to be significant [31]: At a distance of $10\,cm$ and more from the panel, the air flow velocity was never found to be above $10\,cm\,s^{-1}$.

The CFD model's boundary conditions were adapted from the thermal conditions of the laboratory. Note that during the experiment the surface temperature of the panel's frame was higher than the chilled part of the panel. Within the CFD model, the cooling element was simplified and no distinctions were made between the surface temperature of the frame and the chilled part of the panel. As a result, for the CFD simulations the whole panel temperature
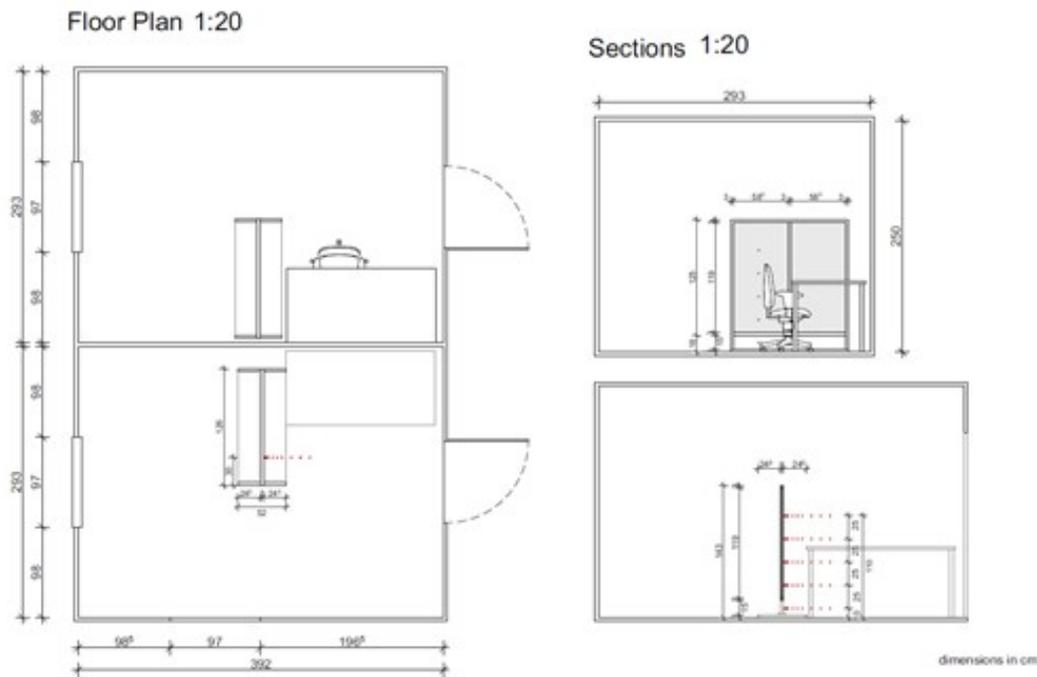
FIGURE 2. Floor plan and sensors locations at the test room.

was considered to be 11 °C.

CFD model was implemented in the finite volume code ANSYS FLUENT 19.0 [32]. The model involves the vertical cooling panel mounted in the test space and a simplified human model for some simulation cases. Ansys design modeler and Ansys meshing were applied as a pre-processor to create the geometry, mesh, and the computational domain. In this model, a mesh with hexahedral element was generated. Buoyancy due to density change was applied in the energy equation. The discrete ordinates (DO) model was applied as the radiation model. The pressure-based solver was applied. The SIMPLE segregated solver was used for pressure-velocity coupling.

Various studies have assessed the performance of turbulence models. For RANS-based models, selecting a proper turbulence model is essential for heat transfer analysis. The answers to the question of the best-performing turbulence models are not always consistent. For instance, the standard k-ε or the RNG k-ε turbulence models are widely implemented for CFD simulations of cooling panel, while SST-K ω model has been applied in other studies. In addition, some studies applied a Laminar flow model. The results of the literature review revealed that the k-ε family of turbulence models presented similar results consistent with laboratory data [33–35]. The SKW and SST turbulence models showed a lesser performance level in predicting air velocity [36, 37]. This disparity can be due to the physical features of the case studies, as well as different computational parameters and settings applied in different studies. Hence, for each specific simulation study, the selection of the turbulence model may have to be reassessed. In the

present study, CFD simulations were carried out using the same geometry but with different turbulence models (RNG k-ε and SST-K ω). To check the reliability of the turbulence model, the simulation results were compared with experimental. Note that we used the discretization scheme employed for the energy, momentum, turbulent kinetic energy, and specific dissipation rate was the second order upwind scheme. Pressure Staggering Option (PRESTO) was used for the pressure discretization scheme.

3. Results and discussion

The air velocity could be an influential parameter for the thermal comfort: Increased airspeed can aid the evaporation of sweat thus leading to a cooling effect, particularly if loose clothing is worn. Nevertheless, too high air velocity may cause discomfort and a draughtiness sensation. Average air speed is typically recommended not to exceed $0.15 \, \text{m s}^{-1}$ in the occupied zone to prevent discomfort [38]. The CFD method can complement the experimental results by allowing the detailed examination of the velocity field at different spots. Given the complexity of the physical processes involved, a candidate simulation model should ideally incorporate complete three-dimensional processes, suitable turbulence models, realistic boundary conditions, and temperature-dependent material properties. In this study, to gain confidence concerning the model's reliability and the employed turbulence model, we first compared the computational results with the laboratory measurements. Subsequently, the study covered the human model analysis.
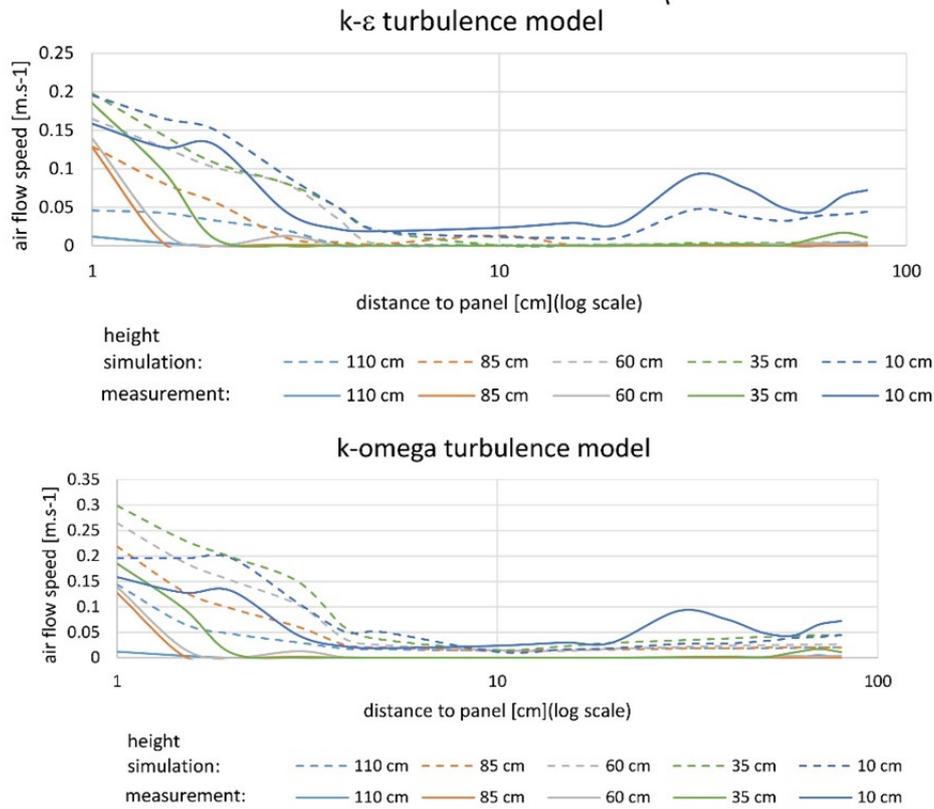
FIGURE 3. Velocity as a function of distance from the panel for two different turbulence models.
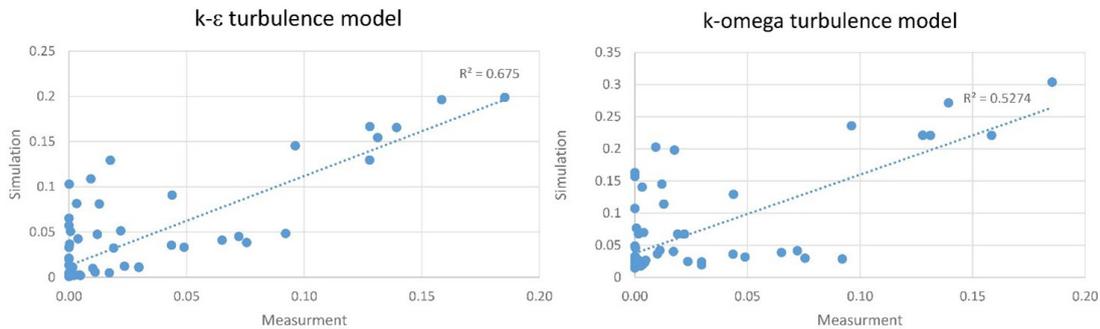

FIGURE 4. Simulated velocities plotted against corresponding measurements for two different turbulence models.

3.1. TURBULENCE MODEL

Two different turbulence models (k-ε and k-ω) were applied for the CFD simulation. During the experiments, air velocity was measured at 75 locations. Figures 3 and 4 illustrates the velocity distribution profile (on the sensors location) obtained from the CFD simulation for two different turbulence models, together with the measured velocity values in the test room. The effect of turbulence models on the air velocity distribution can be seen in this Figure. As such, the k-ε and the k-ω family of turbulence models overestimated the values by up to 0.05 and 0.1 m s^{-1} respectively. In this study, the CFD simulation results obtained while using the k-ε family of turbulence models were found to more closely correspond to the experimental results.

The air velocity values were below the limits of the standards at the assumed seating position of the occupants [38].

As mentioned earlier, in this study we considered a highly simplified human model (body temperature 35) in three distances from the cooling panel (20 cm, 35 cm and 55 cm). The cooling was provided by radiant panels and the manikin acted as the source of heat generation in the room. The flow in the room is dominated by buoyancy. The solver settings were identical to those used for the aforementioned comparison with the experimental results. The main objective of this query was to explore the potential influence of the human presence on the air flow pattern around the radiant cooling panel.

Figure 5 illustrates the simulated velocity contours for the aforementioned simulation scenarios. The results indicate that the velocity in the vicinity of manikin lies in the range of 0.10 to 0.28 m s^{-1}. The velocity range in the zone can be suggested to be
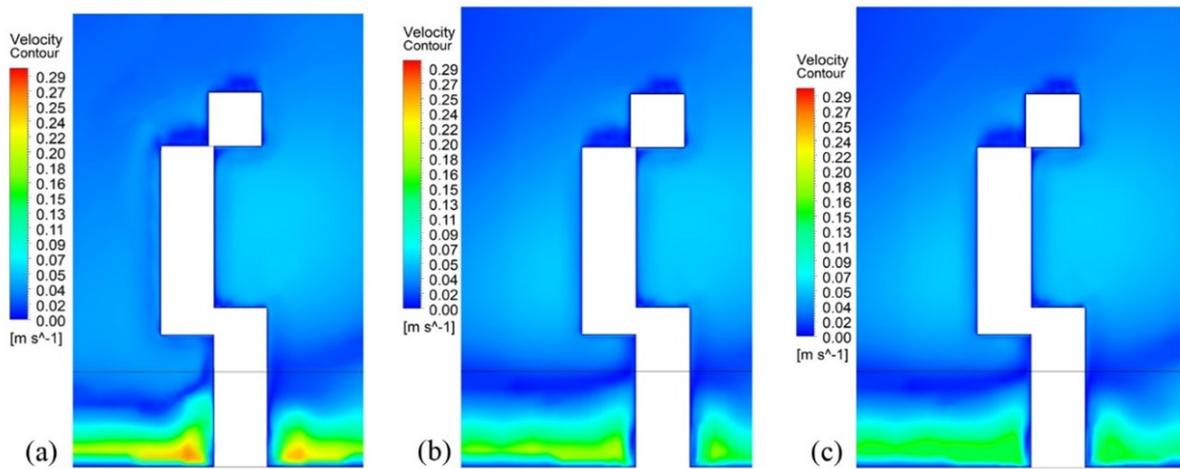
FIGURE 5. Simulated velocity contours around a human model at three distances from the radiant panel, namely distance of human model from panel, namely 20 cm (a), 35 cm (b), and 55 cm (c).

within the recommended maximum velocity range to avoid draft [39]. With increasing distance from the cooling panel, its effect on the velocity field clearly diminishes.

4. Conclusion

Using local radiant cooling panels has been suggested to be a potentially promising alternative to traditional air-conditioning system. Our study involved the thermal analysis of a prototypical local cooling panel installed in a laboratory test room. The thermal performance of the model was evaluated numerically by a commercially available CFD. To gain confidence concerning the model's reliability, its performance was first compared with laboratory measurements. The air velocity fields obtained from the CFD study were found to be acceptable. In addition, the simulated flow around a virtual manikin (simplified human model) at three different locations yielded ranges of velocities than can be suggested to be below the recommended maximum velocity in the test zone.

References

[1] L. Gustavsson, A. Dodoo, R. Sathre. *Climate change effects over the lifecycle of a building – Report on methodological issues in determining the climate change effects over the lifecycle of a building.* Sustainable Built Environment Research Group Linnaeus University, Sweden, 2015.

[2] M. Olonscheck, A. Holsten, J. P. Kropp. Heating and cooling energy demand and related emissions of the German residential building stock under climate change. *Energy Policy* **39**(9):4795–4806, 2011. https://doi.org/10.1016/j.enpol.2011.06.041

[3] J. Liu, S. Zhu, M. K. Kim, J. Srebric. A review of CFD analysis methods for personalized ventilation (PV) in indoor built environments. *Sustainability* **11**(15):4166, 2019. https://doi.org/10.3390/su11154166

[4] J. Niu, J. v. d. Kooi, H. v. d. Rhee. Energy saving possibilities with cooled-ceiling systems. *Energy and Buildings* **23**(2):147–158, 1995. https://doi.org/10.1016/0378-7788(95)00937-X

[5] C. Stetiu. Energy and peak power savings potential of radiant cooling systems in US commercial buildings. *Energy and Buildings* **30**(2):127–138, 1999. https://doi.org/10.1016/S0378-7788(98)00080-2

[6] C. Wilkins, R. Kosonen. Cool ceiling system: A European air-conditioning alternative. *ASHRAE Journal* **34**(8):41–45, 1992.

[7] W.-H. Chiang, C.-Y. Wang, J.-S. Huang. Evaluation of cooling ceiling and mechanical ventilation systems on thermal comfort using CFD study in an office for subtropical region. *Building and Environment* **48**:113–127, 2012. https://doi.org/10.1016/j.buildenv.2011.09.002

[8] K.-N. Rhee, B. W. Olesen, K. W. Kim. Ten questions about radiant heating and cooling systems. *Building and Environment* **112**:367–381, 2017. https://doi.org/10.1016/j.buildenv.2016.11.030

[9] Y. He, N. Li, M. He, D. He. Using radiant cooling desk for maintaining comfort in hot environment. *Energy and Buildings* **145**:144–154, 2017. https://doi.org/10.1016/j.enbuild.2017.04.013

[10] G. Gan. Towards a better indoor thermal environment "CFD analysis of the performance of chilled ceiling systems". In *4th International Symposium on Ventilation for Contaminant Control*, pp. 551–556. 1994.

[11] K. Nagano, T. Mochida. Experiments on thermal environmental design of ceiling radiant cooling for supine human subjects. *Building and Environment* **39**(3):267–275, 2004. https://doi.org/10.1016/j.buildenv.2003.08.011

[12] T. Imanari, T. Omori, K. Bogaki. Thermal comfort and energy consumption of the radiant ceiling panel system: Comparison with the conventional all-air system. *Energy and Buildings* **30**(2):167–175, 1999. https://doi.org/10.1016/S0378-7788(98)00084-X

[13] H. O. Nilsson. Thermal comfort evaluation with virtual manikin methods. *Building and Environment* **42**(12):4000–4005, 2007. https://doi.org/10.1016/j.buildenv.2006.04.027

[14] Q. Chen, K. Lee, S. Mazumdar, et al. Ventilation performance prediction for buildings: Model assessment. *Building and Environment* **45**(2):295–303, 2010. https://doi.org/10.1016/j.buildenv.2009.06.008

[15] T. Hayashi, Y. Ishizu, S. Kato, S. Murakami. CFD analysis on characteristics of contaminated indoor air ventilation and its application in the evaluation of the effects of contaminant inhalation by a human occupant. *Building and Environment* **37**(3):219–230, 2002. https://doi.org/10.1016/S0360-1323(01)00029-4

[16] J. Liu, J. Srebric, N. Yu. Numerical simulation of convective heat transfer coefficients at the external surfaces of building arrays immersed in a turbulent boundary layer. *International Journal of Heat and Mass Transfer* **61**:209–225, 2013. https://doi.org/10.1016/j.ijheatmasstransfer.2013.02.005

[17] P. J. Roache. Quantification of uncertainty in computational fluid dynamics. *Annual Review of Fluid Mechanics* **29**(1):123–160, 1997. https://doi.org/10.1146/annurev.fluid.29.1.123

[18] W. L. Oberkampf, T. G. Trucano. Verification and validation in computational fluid dynamics. *Progress in Aerospace Sciences* **38**(3):209–272, 2002. https://doi.org/10.1016/S0376-0421(02)00005-2

[19] A. Stamou, I. Katsiris. Verification of a CFD model for indoor airflow and heat transfer. *Building and Environment* **41**(9):1171–1181, 2006. https://doi.org/10.1016/j.buildenv.2005.06.029

[20] Y. Li, P. V. Nielsen. CFD and ventilation research. *Indoor Air* **21**(6):442–453, 2011. https://doi.org/10.1111/j.1600-0668.2011.00723.x

[21] J. Liu, M. Heidarinejad, S. Gracik, et al. An indirect validation of convective heat transfer coefficients (CHTCs) for external building surfaces in an actual urban environment. *Building Simulation* **8**(3):337–352, 2015. https://doi.org/10.1007/s12273-015-0212-0

[22] J. Srebric, M. Heidarinejad, J. Liu. Building neighborhood emerging properties and their impacts on multi-scale modeling of building energy and airflows. *Building and Environment* **91**:246–262, 2015. https://doi.org/10.1016/j.buildenv.2015.02.031

[23] S. Murakami, S. Kato, H. Nakagawa. Numerical prediction of horizontal non-isothermal 3-D jet in room based on the k-model. *ASHRAE Transactions* **97**(1):38–48, 1991.

[24] J. A. Myhren, S. Holmberg. Flow patterns and thermal comfort in a room with panel, floor and wall heating. *Energy and Buildings* **40**(4):524–536, 2008. https://doi.org/10.1016/j.enbuild.2007.04.011

[25] T. Kim, S. Kato, S. Murakami, J. Rho. Study on indoor thermal environment of office space controlled by cooling panel system using field measurement and the numerical simulation. *Building and Environment* **40**(3):301–310, 2005. https://doi.org/10.1016/j.buildenv.2004.04.010

[26] N. Gao, J. Niu. CFD study on micro-environment around human body and personalized ventilation. *Building and Environment* **39**(7):795–805, 2004. https://doi.org/10.1016/j.buildenv.2004.01.026

[27] H. Teufl, M. Schuss, A. Mahdavi. Potential and challenges of a user-centric radiant cooling approach. *Energy and Buildings* **246**:111104, 2021. https://doi.org/10.1016/j.enbuild.2021.111104

[28] N. Gao, J. Niu. Modeling the performance of personalized ventilation under different conditions of room air and personalized air. *HVAC&R Research* **11**(4):587–602, 2005. https://doi.org/10.1080/10789669.2005.10391156

[29] M. Kong, T. Q. Dang, J. Zhang, H. E. Khalifa. Micro-environmental control for efficient local cooling. *Building and Environment* **118**:300–312, 2017. https://doi.org/10.1016/j.buildenv.2017.03.040

[30] B. Yang, C. Sekhar. Interaction of dynamic indoor environment with moving person and performance of ceiling mounted personalized ventilation system. *Indoor and Built Environment* **23**(7):920–932, 2014. https://doi.org/10.1177/1420326X13480056

[31] H. Teufl, M. Schuss, A. Mahdavi. Laboratory tests of a prototypical user-centric radiant cooling solution. *Journal of Physics: Conference Series* **2069**(1):012122, 2021. https://doi.org/10.1088/1742-6596/2069/1/012122

[32] Fluent Inc. Ansys Fluent 19.0. 2018 User's guide.

[33] V. Yakhot, S. A. Orszag. Renormalization group analysis of turbulence. I. Basic theory. *Journal of Scientific Computing* **1**(1):3–51, 1986. https://doi.org/10.1007/BF01061452

[34] T.-H. Shih, W. W. Liou, A. Shabbir, et al. A new $k-\epsilon$ eddy viscosity model for high reynolds number turbulent flows. *Computers & Fluids* **24**(3):227–238, 1995. https://doi.org/10.1016/0045-7930(94)00032-T

[35] W. Jones, B. Launder. The prediction of laminarization with a two-equation model of turbulence. *International Journal of Heat and Mass Transfer* **15**(2):301–314, 1972. https://doi.org/10.1016/0017-9310(72)90076-2

[36] F. R. Menter. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal* **32**(8):1598–1605, 1994. https://doi.org/10.2514/3.12149

[37] Q. Chen, W. Xu. A zero-equation turbulence model for indoor airflow simulation. *Energy and Buildings* **28**(2):137–144, 1998. https://doi.org/10.1016/S0378-7788(98)00020-6

[38] ASHRAE. Thermal environmental conditions for human occupancy ANSI-ASHRAE, 1992. In Standard ASHRAE 55-1992.

[39] L. Berglund, A. P. R. Fobelets. Subjective human response to low-level air currents and asymmetric radiation. *ASHRAE Transactions* **93**(1):497–523, 1987.