Proceedings of BS2013:
13th Conference of International Building Performance Simulation Association, Chambéry, France, August 26-28

THERMAL COMFORT AND IEQ ASSESSMENT OF AN UNDER-FLOOR AIR DISTRIBUTION SYSTEM

Bode Florin Ioan[1,2]; **Croitoru Cristiana Verona**[1]; **Dogeanu Angel Madalin**[1]; **Nastase Ilinca**[1]
Organization :[1]CAMBI Research Centre, Technical University of Civil Engineering of Bucharest,
[2]Technical University of Cluj-Napoca, Romania

ABSTRACT

In this article, an under-floor air distribution system is analyzed. The inlet device is placed near a thermal manikin and several turbulence intensities are imposed. Sensitive parts of the body are exposed to the airflow. While global comfort index show no difference in comparison to classical air diffusion strategy, local evaluation of the convective heat flux reveals several particularities. The CFD study gives us not only temperature and velocity fields, but also flow path-lines around a virtual thermal manikin, allowing us to evaluate indoor environmental quality.

INTRODUCTION

When a building is conceived, it must meet two requirements: to be comfortable and functional in accordance with the requirements of the occupants. The building must protect them from adverse external conditions and to provide a pleasant ambience and indoor air quality. Thermal comfort is a subjective term defined by a plurality of sensations and is secured by all factors influencing the thermal condition experienced by the occupant, therefore is difficult to give a universal definition of this concept. Civil engineers have been always challenged to find the balance between thermal comfort and optimal functioning of a building, becoming more and more difficult. We demand high indoor quality, but we ask for sustainability. New solutions are more and more tested and implemented, in order to find better living in a better world.

Maintaining thermal comfort for occupants in buildings in extreme climatic conditioning requirements and irrespective of the environmental outside conditions has been the main focus for the Heating Ventilation and Air Conditioning (HVAC) design engineers and systems developers. We observe, however, that contemporary techniques of air flow diffusions are not optimized simultaneously for these two inseparable goals: thermal comfort and energy savings. This observation can be applied to both buildings and vehicles fields. This paradox is due on one hand to bad diffusion of cold air, and on the other hand, to weakness of the conception of these systems. Behind this, the use of air flow models that are not fully adapted to conditions in the buildings and all other interior spaces can be found. On one hand, in our opinion, this issue finds a theoretical response to the adaptation of existing theoretical models for different indoor (building or other enclosures) conditions, in terms of human thermal comfort.

Nowadays, we have the possibility of using advanced methods and devices both in terms of computing capabilities and experimental techniques. The existing thermal comfort models are all built with simplified assumptions, often limited because of available resources when they were conceived. We have today the opportunity to validate these models by taking into account the variation of several parameters, we also have the opportunity to correct them and to propose new models. On the other hand, a technical answer may come from the conception of the air diffusion devices which have to be optimized for improving mixing between supplied flows and their ambient in order to improve thermal. Nevertheless, this technical direction of research has to be preceded by the theoretical advances in improving the existing comfort models which seem to be inappropriate in many situations [1-4].

In an article from 2001 [5], the Professor Fanger, founder of the first "school" of thermal comfort research and "father" of this scientific field, indicated that the thermal comfort standards are outdated and following their prescriptions cannot lead to acceptable conditions for most users: "We need to reconsider the concept related to our comfort to achieve excellence in environmental quality. Our goal should be essential to provide fresh air, accompanied by a pleasant feeling, refreshing, without any adverse health effect and a comfortable thermal environment for all users." said the Professor in [5].

In this context, this study is a part of a larger experimental and numerical campaign which is intended among other directions to study the influence of the turbulence intensity at the exit plane of the terminal air diffusion devices on the local draft sensation and thermal discomfort of mixing and personalized ventilation users.

To what extent the turbulence intensity of the flows generated by various air diffusion devices can affect the comfort and what are the consequences of an "incomplete" assessment based on existing models? How is affected the design of ventilation and air conditioning systems due to the use of these models for pre-evaluation of interior parameters?

The UFAD (Under-Floor Air Distribution) diffusion strategy has become more and more interesting and gained popularity in the new trends, but the problems are not yet clearly pointed. The literature reveals a lot of interest in evaluating the energy efficiency of the UFAD systems [6, 7], but this direction only can lead to complaints for the occupants, from the thermal comfort point of view. In this article, we evaluate several comfort parameters and indices for an under-floor ventilation system. Is the global thermal comfort comfortable for everybody?

In several articles, different UFAD systems are evaluated [8-10] and the thermal comfort of the occupants is discussed in detail, but this method is limited to the use of well-known indices.

EMPLOYED METHOD

Numerical case study

Numerical simulations by CFD approach using a RANS (Reynolds Averaged Navier Stokes) model were performed to study the airflow and heat transfer around a human body for three values of turbulence intensity at the jet inlet (see Table 1). The virtual thermal manikin was placed in a test cell (Fig. 1) which is a reproduction of a real laboratory facility [11]. Figure 1 gives the dimensions of the test cell, the position of the thermal manikin and the inlet and outlet devices. The outlet on the floor and the two outlets in the upper part of the room are colored in red.

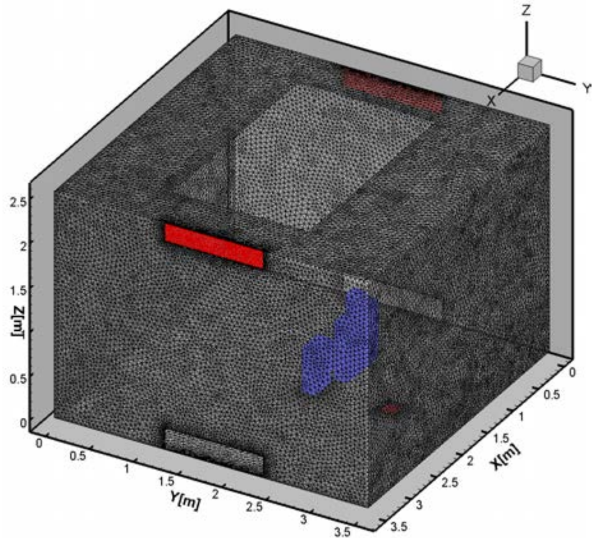


Figure 1 The test cell with the thermal manikin inside

The other two openings from the lower part of the room and the one from the ceiling are part of the real experimental facility but there were not used in this numerical simulation.

The velocities in the occupancy zone and the noise level must be under certain limits. In this case, the inlet is positioned directly in the occupied zone, with low air velocity and temperatures. Hot and polluted air is evacuated in the upper part of the room. The efficiency of the ventilation system depends on the interior configuration. The inlet airflow is around 400 m^3/h. The studied inlet turbulence conditions are 2%, 10% and 30%, for an inlet velocity of 0.5 m/s.

The accuracy of a CFD simulation depends, in a high percentage, on the way of reproducing the geometry that defines the calculation domain and the heat sources. The virtual manikin is in a sitting position and has a height of 1.2 m and a body surface of 1.95 m2. This way, the virtual manikin has 8 distinctive segments which can be "analyzed" separately (head, torso, back, left arm, right arm, upper legs, upper legs back and legs). In the meshing process, a viscous layer was inserted over the surface of the human, using 4 layers with a first layer thickness of 1.5 mm and a growth factor of 1.2 (Figure 2) [12]. Walls were also provided with a viscous layer of four layers with a first layer thickness of 6mm and a growth factor of 1.2. The result was a mesh with 4.5 million cells, refined enough in the in the interest zone were we suspect large gradients of temperature and velocity, which was imported in Fluent 14.

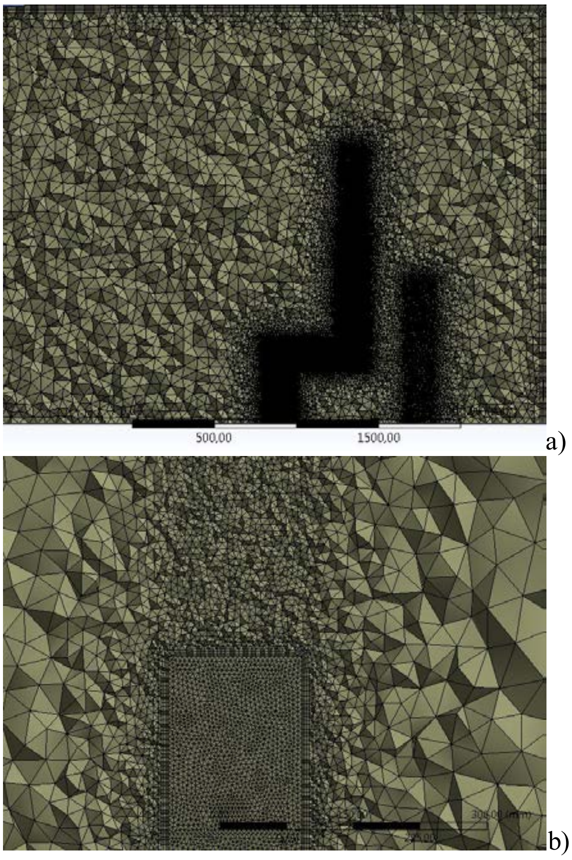


Figure 2 a) The test cell mesh b) Detail of the mesh around the manikin

A mesh dependency study was conducted and showed that this grid was fine enough in order to obtain stable results.

The boundary conditions used during the numerical study considered the walls being at a constant temperature of 26°C. The surfaces of the different segments of the virtual manikin were considered as having temperatures that were previously determined on a thermal manikin using an infrared camera (see Table 1).

Table 1
Temperature boundary conditions for the CFD study

BODY SURFACE				*Walls*
Head	*Torso and back*	*Arms*	*Legs*	
34°C	32°C	30°C	27°C	26°C

The manikin is considered to be sited at a desk, so that the interior thigh is placed on a chair which has the air temperature, 26°C. The inlet device is placed at 20 cm behind the manikin, right on the floor surface. The inlet temperature is 22 °C, usual situation of air cooling.

A "*second order discretization*" method was used for the calculation of convective terms and the SIMPLE algorithm for pressure-velocity coupling. The chosen turbulence model was k-epsilon realizable [13] with low Reynolds number correction. Convergence solution is assumed to be achieved when the dimensionless residuals of the flow equations are less than 10^{-3}.

RESULTS AND DISCUSSION

In the design of indoor environment, it is still not acknowledged that convection flows caused by heat sources like the human body plume may significantly affect the flow distribution in rooms [14]. Generally, attention is given only on the flow generated by the air diffusion terminal devices. As shown by Kosonen et al [14] the point of occurrence of the maximum air velocity in the occupied zone depends on the heat source strength and its distribution in the room. Thus, the air flows interaction in ventilated rooms is of great importance when estimating occupants' comfort. In this context, the second goal of this campaign is to take into account the presence of the human body thermal plume in order to obtain realistic conditions in both experimental and numerical investigations of air distribution in ventilated rooms. The thermal comfort assessment is often evaluated with PMV model, presented in Figure 3. We also used this model for thermal comfort evaluation in our test cell.

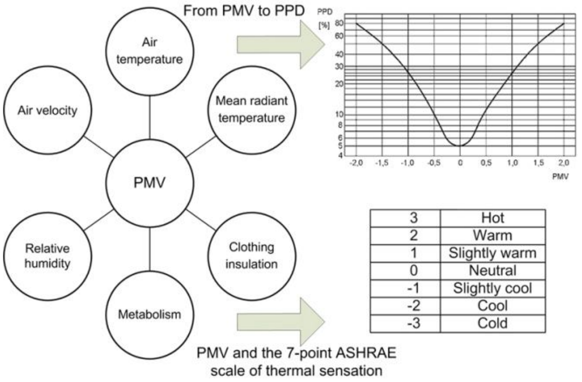


Figure 3a) PMV-PPD diagram [15]

On the other part, as it has been shown by Fanger [16], the velocities and the turbulent characteristics of the flows may generate a thermal discomfort translated by the sensation of "draught" as "an undesired cooling of the human body caused by air movement" [16, 17].

This way, we wanted to check first, the influence of the variation of the jet initial turbulence intensity on the behavior of the global temperature and velocity fields inside the test cell. Therefore, in Figure 4 we are presenting the velocity fields in sagittal planes for three values of the turbulence intensity (Tu).

We can observe that the thermal plume is dissipated with the raise of the turbulence intensity, as well as the velocity inside the room.

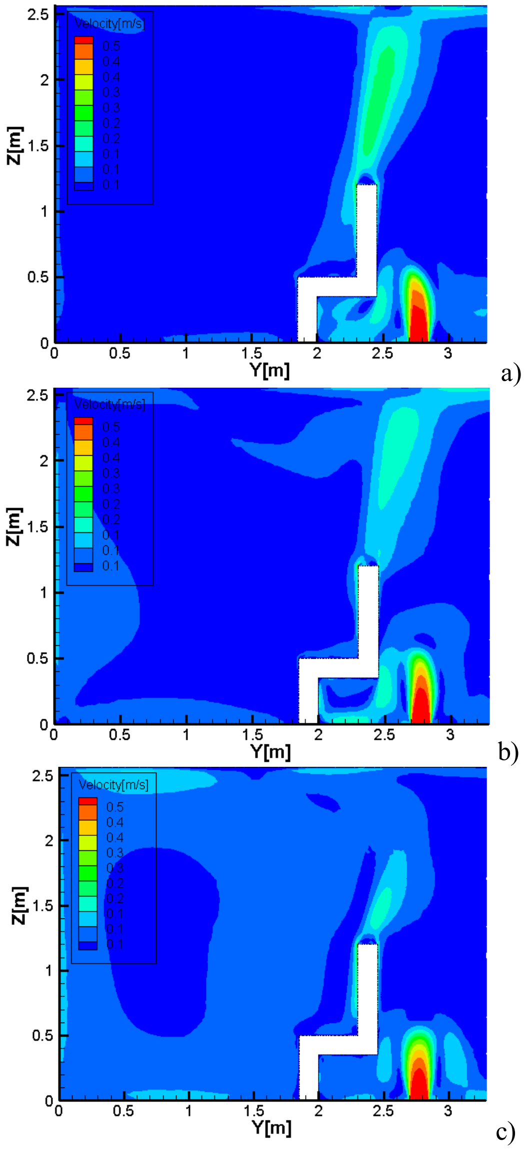


Figure 4 The velocity distribution in the sagittal plane a)2% Tu b) 10%Tu c)30% Tu

The CFD simulation results allow us to easily evaluate the PMV and DR index as defined by Fanger. These differences are not captured by classical comfort indices as Draft and PMV [18]., as the Figure 5 shows. The DR index has three ranges: 0, 15 and 30 and the PMV scale is the usual seven point scale: -3...+3 While these indices should be sensitive to velocity and temperature fluctuations, we couldn't find any variation with of their values with the turbulence intensity imposed at the several studied air diffusers. *The assessment made by classical indices is indicating a neutral thermal environment.*

This conclusion can be easily seen in the following figure:

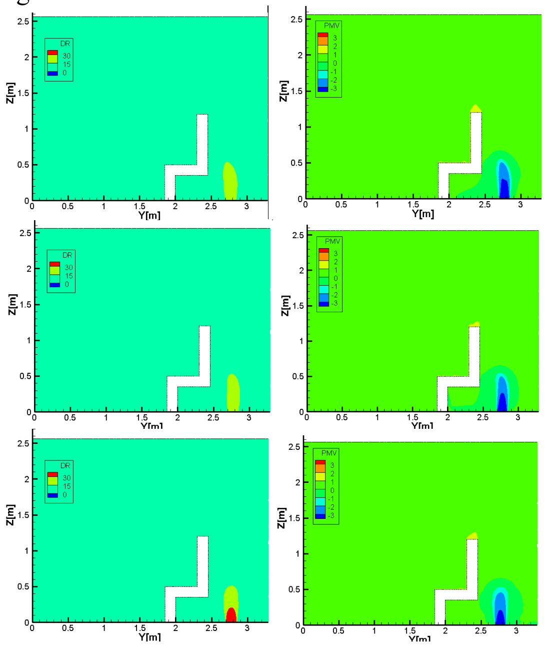


Figure 5 The DR(left) and PMV(right) distribution in the sagittal plane a)2% Tu b) 10%Tu c)30% Tu

If we evaluate the thermal comfort through another perspective, we observe that the convective heat flux released by the manikin's body parts shows changes between the three cases. The heat flux released decreases with the amplifying turbulence intensity.

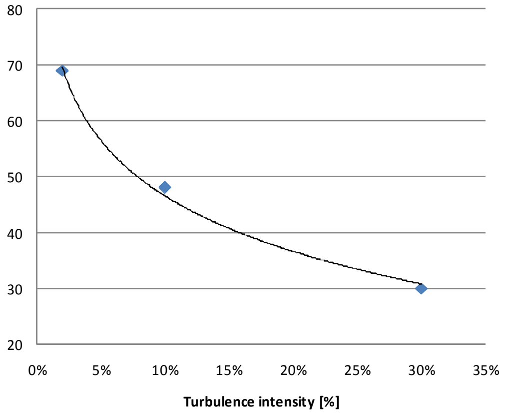


Figure 6 Variation of maximum convective heat flux on the back of the manikin in function of the inlet turbulence intensity

If we evaluate the back of the manikin (one of the parts that are likely to be in local discomfort because of the inlet positioning), we can observe that the local heat flux is different.

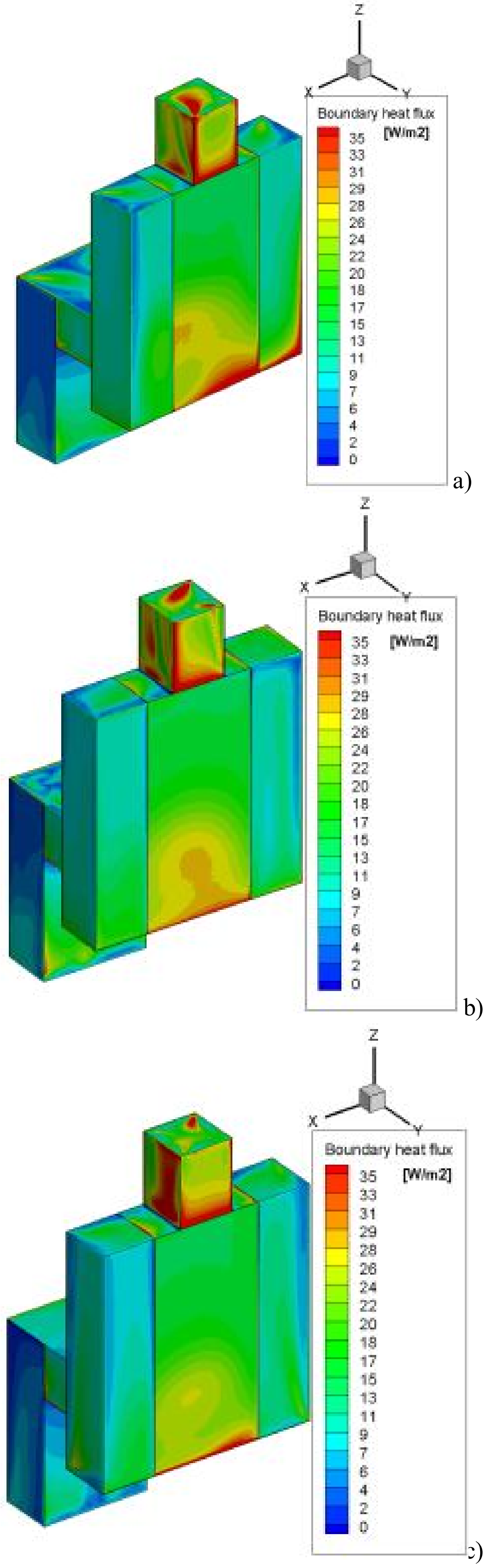
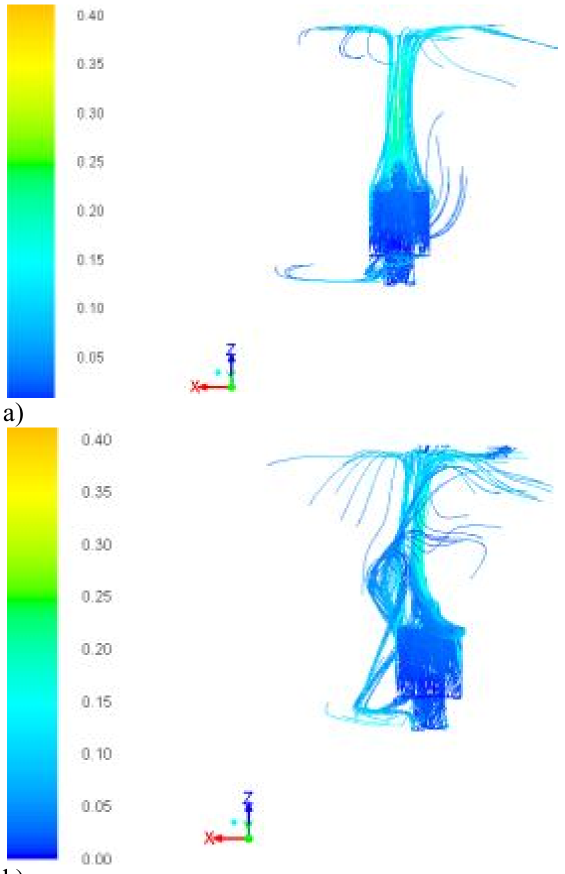


Figure 7. Variation of convective heat flux on the back of the manikin a)Tu= 2% b)Tu=10% c)Tu= 30%

The maximum value on this surface is decreasing with the inlet intensity. This tendency can be observed on this surface by plotting the convective heat flux. The finding leads us to the idea of further study on evaluating the thermal comfort in relation with other parameters and using new approaches.

An important problem that might appear for this under-floor ventilation is the fact that the dust and the particles found naturally on the ground are led to the breathing zone of the occupants. Can we obtain by this type of ventilation strategy an optimal environment? If yes, on what conditions?

The last figure shows the path lines of the airflow, colored by the velocity of particles. The air mixing is more pronounced in the last case of Tu=30%, which can lead to the conclusion that particles are brought faster in the breathing zone for this last case.



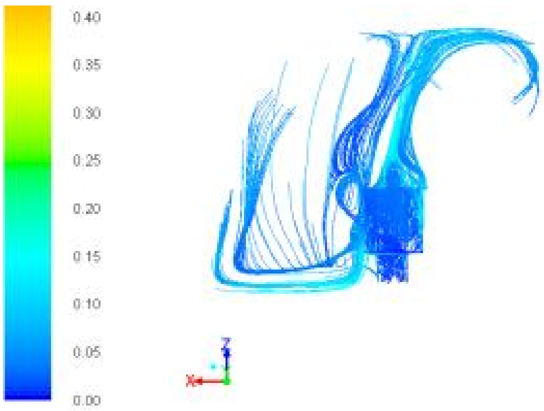


Figure 7. Path lines of air distribution colored by velocity magnitude a)Tu= 2% b)Tu=10% c)Tu= 30%

CONCLUSION

This paper is a short numerical study of the influence on human comfort of inlet air turbulence intensity in a case of under-floor ventilation system.

While classical indices show an uniformity in the room, new evaluation methods indicate a variation of local conditions that can lead to discomfort for certain body parts.

We obtain a more uniform environment for the case with Tu=30%. On one hand, a higher turbulence intensity lead to lower heat flux releases for the exposed zones of the human body parts, fact that might reduce the local discomfort risk On the other hand, the analysis of air distribution inside the test cell reveals that an UFAD would be better used with low inlet turbulence intensities in order to avoid mixing air with dust particles.

NOMENCLATURE

Tu = Air turbulence intensity [%]

ACKNOWLEDGEMENT

This work was supported by the grants of the Romanian National Authority for Scientific Research, CNCS – UEFISCDI, project numbers: PN-II-ID-PCE-2011-3-0835 and PN-II-PD-PCE-2011- 3-0099.

REFERENCES

1. Croitoru, C., et al. *Numerical and experimental modeling of airflow and heat transfer of a human body*. in *Roomvent 2011*. 2011. Trondheim, Norway.
2. Croitoru, C., et al., *Inlet turbulence intensity influence on the thermal comfort in the case of a mixing ventilation system* submitted to Building and Environment, 2011.
3. Alahmer, A., et al., *Vehicular thermal comfort models; a comprehensive review.* Applied Thermal Engineering, 2011. **31**(6–7): p. 995-1002.
4. Cheng, Y., J. Niu, and N. Gao, *Thermal comfort models: A review and numerical investigation.* Building and Environment, 2012. **47**(0): p. 13-22.

5. Fanger, P.O., *Human requirements in future air conditioned environments* International Journal of Refrigeration, 2001. **24**: p. 148-153.
6. Alajmi, A. and W. El-Amer, *Saving energy by using underfloor-air-distribution (UFAD) system in commercial buildings.* Energy Conversion and Management, 2010. **51**(8): p. 1637-1642.
7. Lee, K.H., et al., *Thermal decay in underfloor air distribution (UFAD) systems: Fundamentals and influence on system performance.* Applied Energy, 2012. **91**(1): p. 197-207.
8. Zukowski, M., *A new formula for determining a minimum recommended value of inlet air velocity from UFAD system to prevent occupants from draught risk.* Building and Environment, 2007. **42**(1): p. 171-179.
9. Zukowski, M., *Modeling and designing heating and ventilation system with underfloor air distribution.* Energy and Buildings, 2006. **38**(6): p. 600-609.
10. Chung, J.D., H. Hong, and H. Yoo, *Analysis on the impact of mean radiant temperature for the thermal comfort of underfloor air distribution systems.* Energy and Buildings, 2010. **42**(12): p. 2353-2359.
11. Nastase, I., et al., *Lobed grilles for high mixing ventilation - An experimental analysis in a full scale model room.* Building and Environment, 2011. **46**(3): p. 547-555.
12. Sorensen DN, V.L., *Modeling airflow and heat transfer around a seated human body by computational dynamics.* Building and Environment 2003. **38**(6): p. 753-762.
13. Makhoul, A., K. Ghali, and N. Ghaddar, *Thermal comfort and energy performance of a low-mixing ceiling-mounted personalized ventilator system.* Building and Environment, 2013. **60**(0): p. 126-136.
14. Kosonen, R., et al., *Impact of heat load location and strength on air flow pattern with a passive chilled beam system* Energy and Buildings, 2010. **42**(1): p. 34-42.
15. van Hoof J Fau - Mazej, M., J.L.M. Mazej M Fau - Hensen, and J.L. Hensen, *Thermal comfort: research and practice.* (1093-4715 (Electronic)).
16. Fanger, P.O., et al., *Air turbulence and sensation of draught.* Energy and Buildings, 1988. **12**(1): p. 21-39.
17. Fanger, P.O., *The new comfort equation for indoor air quality.* Ashrae Journal, 1989. **31**(10): p. 33-38.
18. Fanger, P.O., *Thermal comfort*. 1982, Malabar, FL.: Robert E. Krieger Publishing Company.