AIR TURBULENCE INTENSITY INFLUENCE ON THE THERMAL COMFORT EVALUATION FOR DIFFERENT VENTILATION STRATEGIES

Cristiana Verona Croitoru[1], Ilinca Nastase[2*], Florin Bode[3]

1 CAMBI Research Center
Technical University of Civil Engineering in Bucharest, Building Services Faculty, Avenue Pache Protopescu 66, Bucharest, Romania

2 CAMBI Research Center
Technical University of Civil Engineering in Bucharest, Building Services Faculty, Avenue Pache Protopescu 66, Bucharest, Romania
**Corresponding author: ilinca.nastase@cambi.ro*

***3** Technical University of Cluj-Napoca, Mechanical Engineering Department,B-dul Muncii nr.103-105, D03, Cluj-Napoca, Romania*
CAMBI Research Center, Technical University of Civil Engineering in Bucharest, Building Services Faculty, Avenue Pache Protopescu 66, Bucharest, Romania

ABSTRACT

Thermal comfort is a subjective term, closely related to the sensation of warm or cold for the occupants, defining the state of mind of humans that expresses satisfaction with the surrounding environment. Since we spend more than 90% of our time inside buildings or vehicles, achieving a good thermal quality of this enclosed environment is vital. An optimal thermal comfort prediction can lead to „bien-être", efficiency in our work, unaltered health and even energy economy.
Among the newest and highly used approaches for thermal comfort studies are CFD methods, giving the possibility of extended parametric studies and in depth analysis of every thermo-fluidic parameter involved.
In this paper, several ventilation strategies were tested, in order to evaluate the thermal comfort of the occupants. We are comparing two mixing ventilation, and two displacement ventilation cases. This study is a part of larger experimental and numerical campaign intended to evaluate the influence of several flow parameters, such as the turbulence intensity at the inlet of the terminal air diffusion devices, on the local draft sensation and thermal discomfort of ventilation users. A numerical thermal manikin was placed in the centre of a test cell where different strategies of ventilation can be used. The results of CFD simulations were validated in a previous experimental campaign where a thermal manikin was used. While the PMV, and the PPD indexes seem to be less sensible to the value of the local turbulence intensity, the heat fluxes exchanged between the different body parts and their environment are highly varying between the studied cases. Local correlations between the turbulence intensity and convective heat flux of the body were found.

KEYWORDS
Thermal comfort; CFD; turbulence intensity; ventilation strategies;

INTRODUCTION

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 – over 30 years ago for the most used. 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 [5-8].

In an article from 2001 [3], 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 [3]. At the same time if we consider two bibliographic articles at a distance of 20 years - [4] and [5] – we can see that nothing has changed in definition and use of these models and evaluation indexes of interior ambiental comfort.

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.

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

EMPLOYED METHOD

Numerical case study

Numerical simulations using a CFD approach using a RANS (Reynolds Averaged Navier Stokes) model were performed to study the airflow and heat transfer around a human body for several air diffusion strategies and for different values of the turbulence intensity at the jet inlet (see Table 1). The virtual thermal manikin was placed in a test cell (Fig. 1a) which is a reproduction of a real laboratory facility [10]. Figure 1a, gives the dimensions of the test cell and position of the thermal manikin. In Figure 1b are presented different inlet and outlet conditions studied through numerical simulation corresponding to the four cases, marked below and where the inlets are coloured with blue and the outlets with red. We define two virtual median plans of the manikin: - the median plane, passing through the heart (in green in Fig. 1 b) – **coronal plane**, and the transverse median plan, symmetric – **sagittal plane**.

Case 1 and Case 2 (**M1** and **M2)** are mixing ventilation distributions where the inlet device is placed at the upper part of the room and the outlet at the lower part (M1) and, vice-versa, the inlet at the lower part and the outlet at the upper part of the room respectively (M2). Case 3 **(D1)**, represents the simulation of a displacement ventilation strategy, the inlet device being considered as the entire duct surface, indicated with yellow in Fig. 1 a and in blue in Fig. 1c. For Case 4 **(D2)**, we considered the inlet on the whole front wall. Mixing ventilation implies supplying an air jet with relatively high velocity and high air change rates. The velocities in the occupancy zone and the noise level must be under certain limits. In the displacement 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. Piston-type ventilation strategy is a particular case of displacement ventilation, where the whole surface of a wall is used as an inlet device. The principal demand is to have low velocities and turbulence intensities. For our case, the air exchange rate is over 200 h^{-1}. This strategy is applied for white chambers, operating rooms etc. [13] . All studied inlet velocity and turbulence conditions are given in Table 1

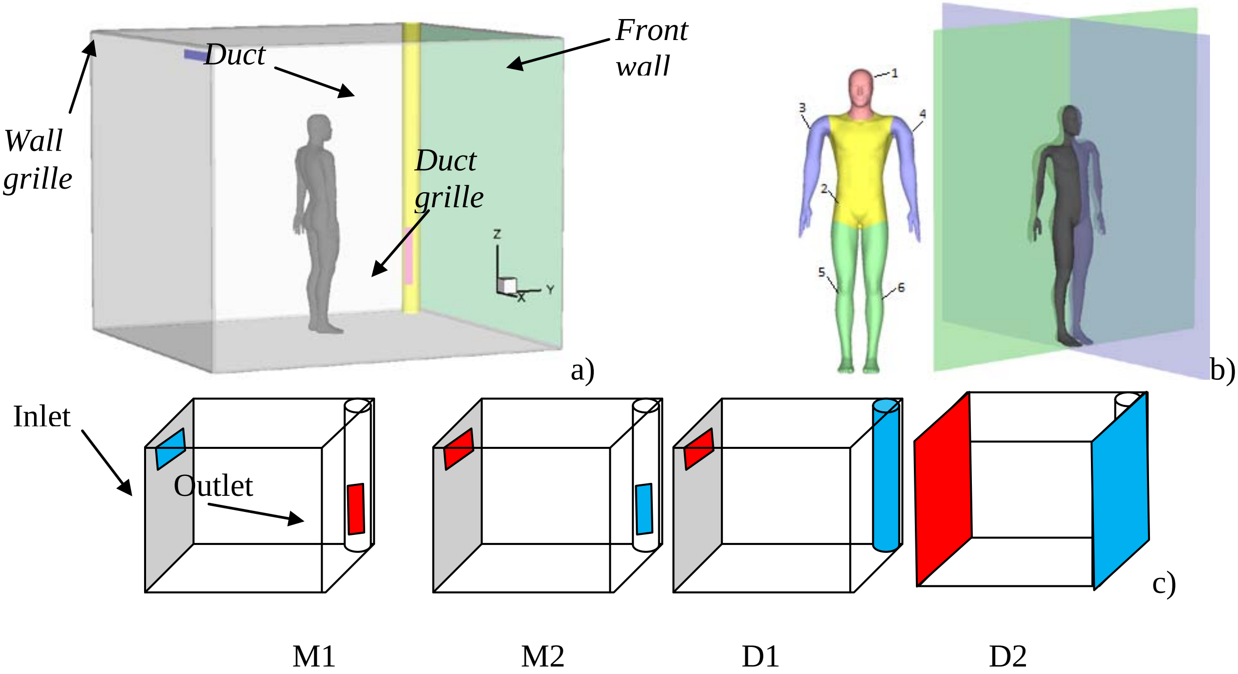

Fig.1: a) Possible inlets in the test celle, b) Human body parts: 1- Head, 2- Torso, 3-Right arm, 4-Left arm, 5-Left Leg, 6-Right Leg, sagittal plan (purple) and coronal (green)c) Inlet and Outlet definition for the four studied cases

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 approach used here is based on freely available geometry from the MakeHuman project [11] that was meshed in an automatic unstructured grid generator HEXPRESS from Numeca [12]. The virtual manikin has a height of 1.88 m and a body surface of 1.9 m^2. This way, the virtual manikin has 18 distinctive segments which can be “controlled” separately. However in this study we grouped the segments by three in order to obtain larger regions which are similar to the ones of the experimental thermal manikin. Three segments from one region have the same number in Fig.1b (i.e. head, body, arms and legs). The meshing process consisted of 5 steps: initial mesh generation, mesh adaptation, mesh snapping, mesh optimization and viscous layer insertion. During the final step, a viscous layer was inserted over the surface of the human, using 20 layers with a first layer thickness of 0.2 mm and a growth factor of 1.13 (Figure 1 c) [13]. The result was a mesh with a total of 2.2 million cells which was imported in Fluent 12. 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 20°C. The surfaces of the different segments of the virtual manikin were considered as having temperatures which were previously determined on a thermal manikin using an infrared camera (see Table 2).

Ventilation strategy	Inlet position	Imposed inlet velocity	Imposed turbulence velocity	Inlet type	Outlet type
M1	• Upper part-wall grille	2 m/s	0%, 3%, 10%, 30%, 50%	Velocit y inlet	Pressure outlet
M2	• Lower part-duct grille	1 m/s	3%, 10%, 30%		
		2 m/s	3%, 10%, 30%		
D1	• Duct	0.3 m/s	3%, 10%, 30%		
D2	• Front wall	0.2 m/s	0%, 3%, 10%, 30%, 50%		
		0.3 m/s	0%, 5%, 3%, 10%, 15%, 20%, 25%, 30%, 50%		

Table 1: Studied cases: velocity and turbulence intensity values imposed at the inlet and other boundary conditions

Body surface				*Walls*
Head (1)	Body (2)	Arms (3, 4)	Legs (5, 6)	
34.2°C	31.9°C	30°C	26.8°C	t=20°C

Table2: Temperature boundary conditions for the CFD study

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-ω standard [14] with low Reynolds number correction. The radiation flux was calculated by using the "surface-to-surface" model which allows setting the radiation surfaces. Convergence solution is assumed to be achieved when the dimensionless residuals of the flow equations are less than 10^{-3}.

Experimental validation

In this study, Particle Image Velocimetry measurements were used for validation. They targeted experimental validation of the velocity distribution obtained by numerical study of convective flow above the manikin's head and of the airflow released by an inlet device from a classical mixing ventilation case. The validation test consisted in compare a mean field of the velocity magnitude from PIV measurements (the employed PIV system is described in [10]) in the sagital plane above the manikins head, and the numerical velocity field, in the case M1. In Fig.3 we compared the two components of velocity fields W and U and the plan distribution of velocity magnitude obtained from PIV measurements and numerical simulations using the k-omega SST model. In this figure, we can see the existence of a smaller convective flow in the direction of the forehead, which meets the main convective flow. This secondary flow seems to meet with another flow coming from behind the manikin. General shape of the three fields is similar and we emphasize that k-omega SST model satisfactorily reproduces the dynamics of convection plume.
Even if the stagnation region is more extended in the experimental case, the two contour plots display some common characteristics: the presence of a smaller flow coming along the forehead which is reuniting with a stronger plume above the head. This latter flow seems to be connected with another plume coming from the back of the manikin. The values of the velocity magnitude from both numerical and experimental fields are very similar to the ones presented by Sorensen and Voigt [2].

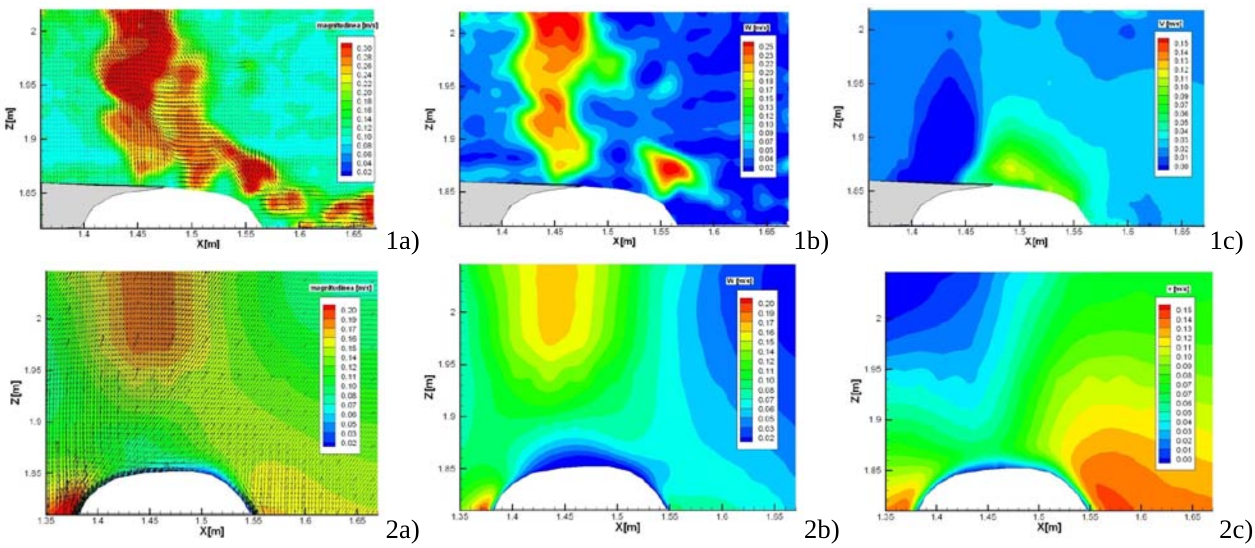

Fig 3. Velocity field comparison –sagittal plan- with jet (Tu=10%): 1 – experimental, 2 –numeric
a) Velocity magnitude [m/s] b) W component [m/s] c) U component [m/s]

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 [1]. Generally, attention is given only on the flow generated by the air diffusion terminal devices. As shown by Kosonen et al [1] 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.

As it has been shown by Fanger [14], 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" [14, 15]. This way, we wanted to check first, the influence of the variation of the jet initial turbulence intensity on the behaviour of the global temperature and velocity fields inside the test cell. Therefore, in Fig. 4 and 5 we are presenting the temperature and velocity fields in both coronal and sagital planes for all studied cases, for two values of the turbulence intensity.

In the mixing ventilation cases, M1 and M2, the air surrounding the manikin seems to be well mixed, allowing nevertheless the observation of the thermal plume. Slight differences between temperature or velocity fields, for the two initial turbulence intensities might be observed, especially in the region of the convective plume flow. The turbulence intensity value seems to influence more the velocity and temperature distributions for the displacement ventilation case D1. As for the piston case D2, there is no obvious influence of the turbulence intensity on the fields in Fig. 4 and 5.

The CFD simulation results allow us to easily evaluate the PMV index as defined by Fanger. In Fig. 6 are given the corresponding distributions of the global PMV and PPD of the room for each studied case and each turbulence intensity level. While these indexes 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.

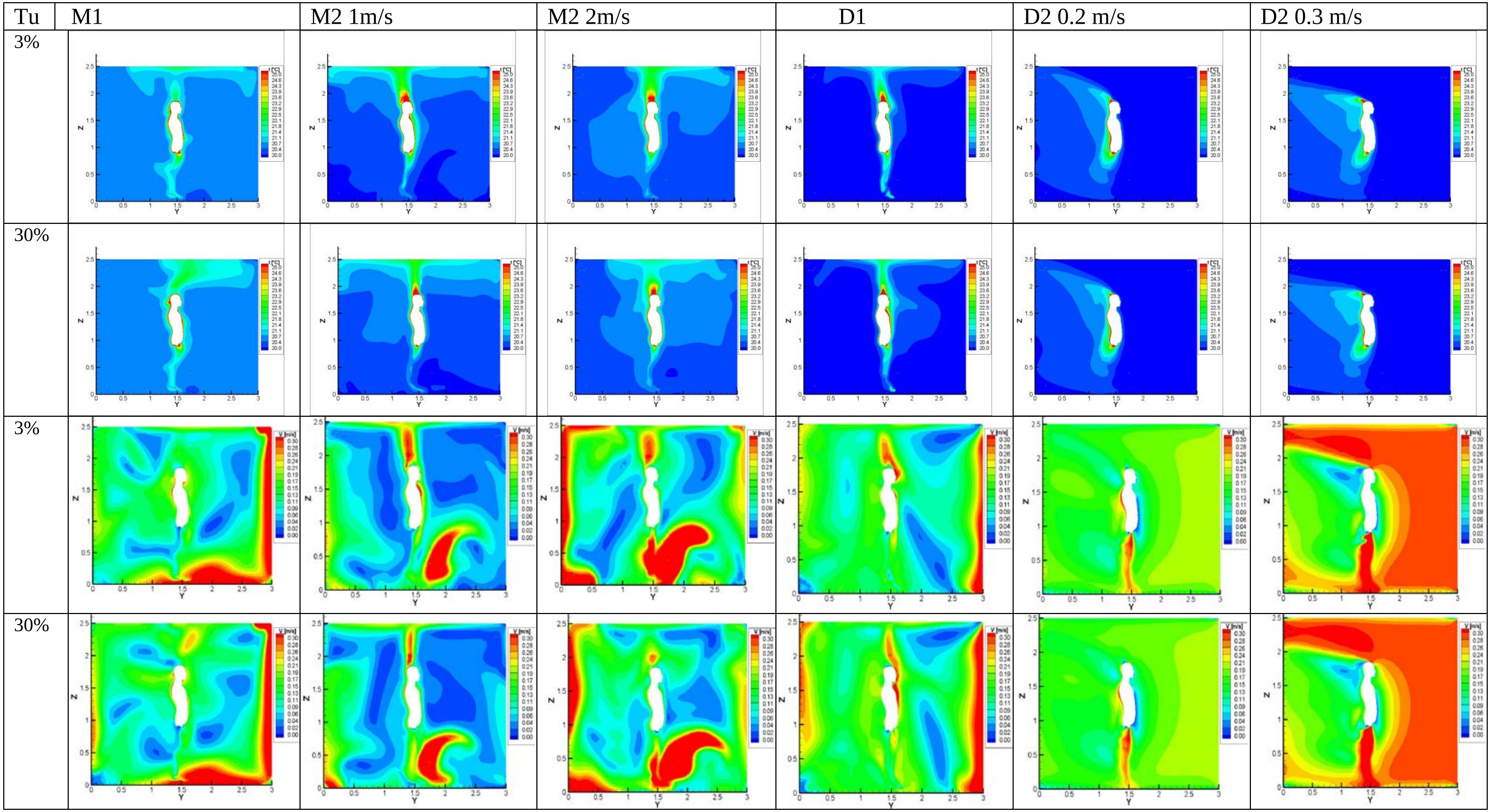

Fig.4: Temperature and velocity fields in the sagital plane for Tu=3% and Tu=30% for all studied cases

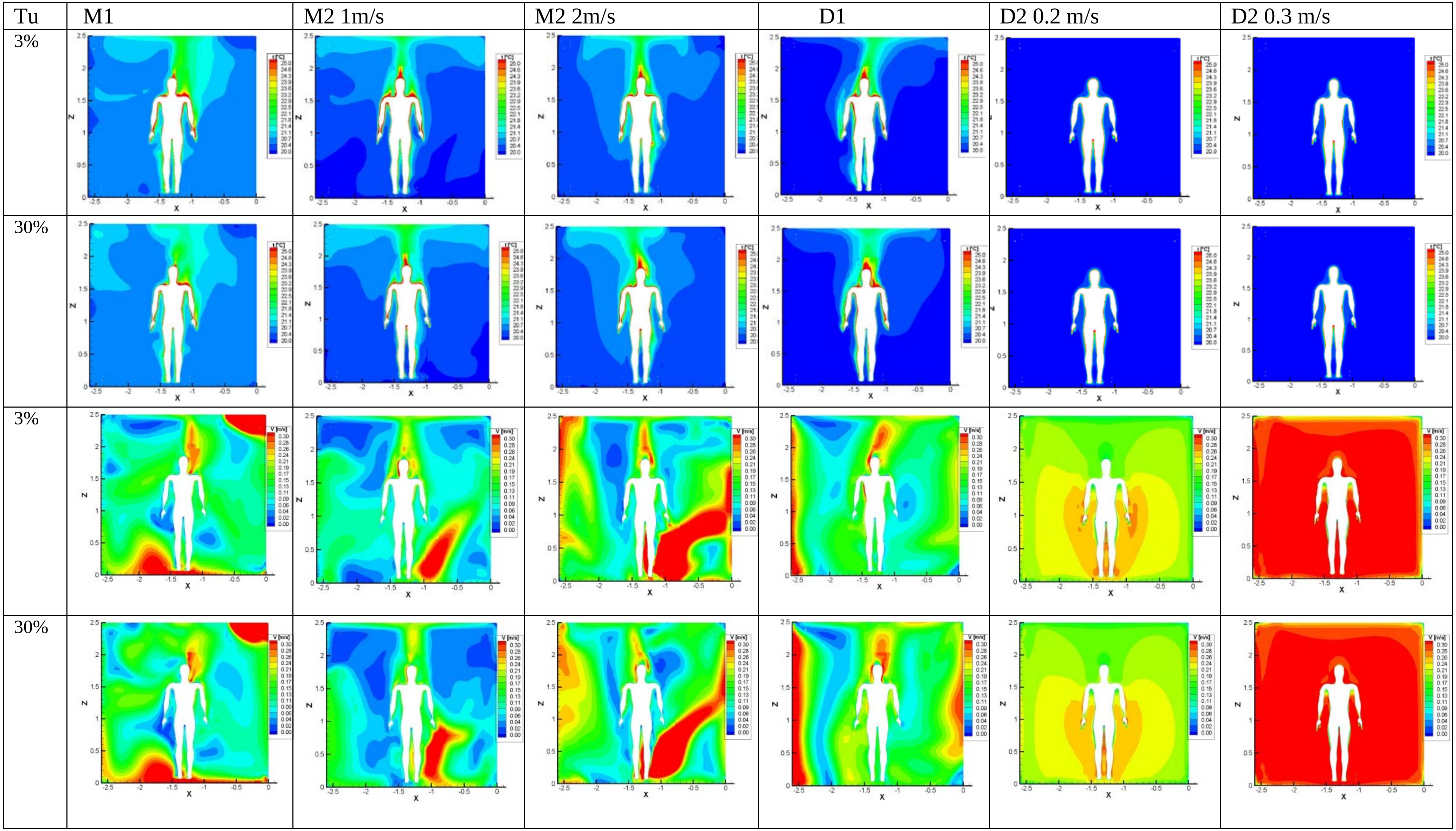

Fig.5: Temperature and velocity fields in the coronal plane for Tu=3% and Tu=30% for all studied cases

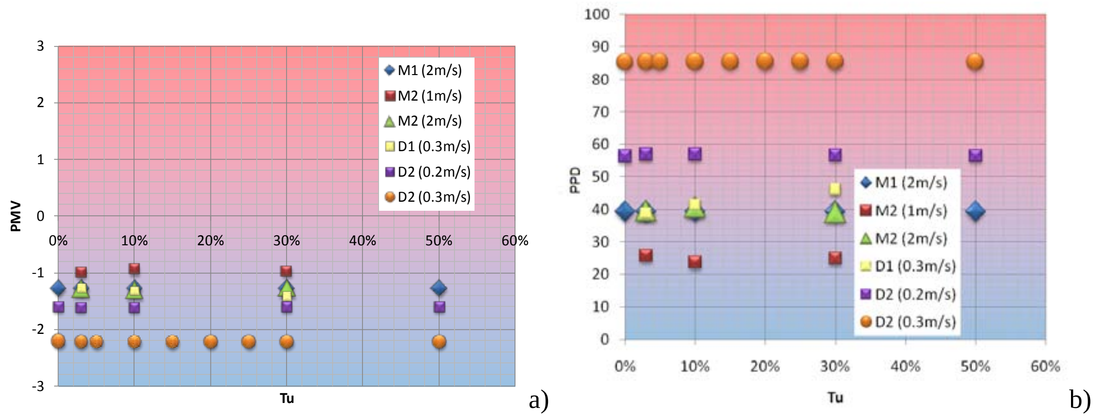

Fig. 6: Global PMV(a) and PPD (b) of the room for each studied case

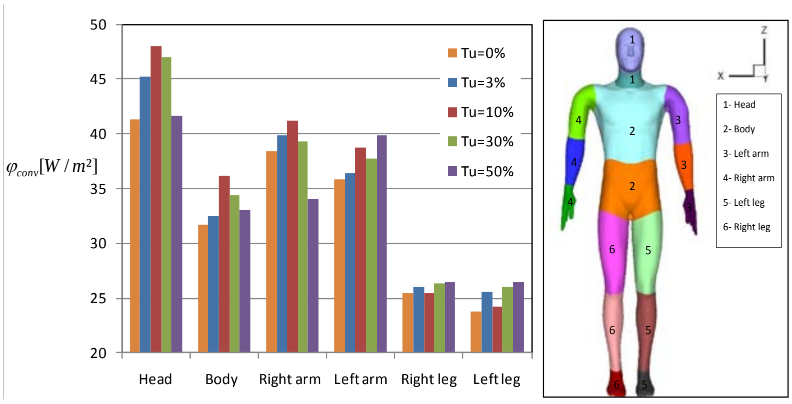

Fig. 7: Average convective fluxes of each body part for M1 case

Another way of studying the thermal comfort implications in changing the initial turbulence level at the inlet of the jet flow is looking to the heat transfer between the manikin's body and its environment. This way we represented in Fig.7 average convective fluxes for different body segments, for the M1 case. From both figures it can be observed that a more intense heat transfer occurs for the head and **body** of the manikin in the case having the initial turbulence intensity *Tu=10%*. As the fluctuations of the convective heat flux for different parts of the body (Fig.7) are quite important with the variation of the initial turbulence intensity at the jet inlet, they have to interfere with the local thermal comfort of the body. In the same time we saw that the PMV and PPD values are not sensitive to these fluctuations.

As it is obvious that the virtual body suffers a non-uniformly distributed convective heat transfer, with high variations between the different studied cases, it is necessary to employ another quantity being able to quantify these variations. A thermal comfort index employed in nonuniform thermal environments is the equivalent temperature t_{eq} which takes into account the combined effect of the local air temperature, local thermal radiation, local air velocity based on the local heat transfer rate at the skin surface [24]. The charts of the body parts sensations in Fig. 8, are showing strong differences between the considered turbulence intensities values in the case M1.

Given this dynamics of the convective fluxes we wanted to get insight into the local phenomena governing the thermal transfer occurring at local scales. Indeed, if we question ourselves which is the level of the local turbulence intensity influence on the thermal comfort and thermal transfer, the most appropriate approach would be to search possible correlations between the local turbulence intensities and local convective fluxes.

Figure 9 is an example of such type of correlation that can be done between the turbulence intensity values and the local convective heat flux. As we can see in Fig. 9, in the region behind the legs, the convective heat flux is in the range of 25 W/m to 48 W/m, for all the mean turbulence intensities around the legs. For each case we observe an increase of approximately 14% on convective heat flux, with a maximum of 18% for the case mixing ventilation with inlet at the lower part of the room (Case 2B). The fact that the turbulence intensity has an important impact on heat transfer between the body and the environment is well known, but this hypothesis has never been quantified before in the literature. *We can observe that only by varying the inlet imposed turbulence intensity of the air we obtain important differences by quantifying the thermal effect.* This conclusion opens new research directions in which this parameter is seriously taken into consideration for evaluating thermal comfort.

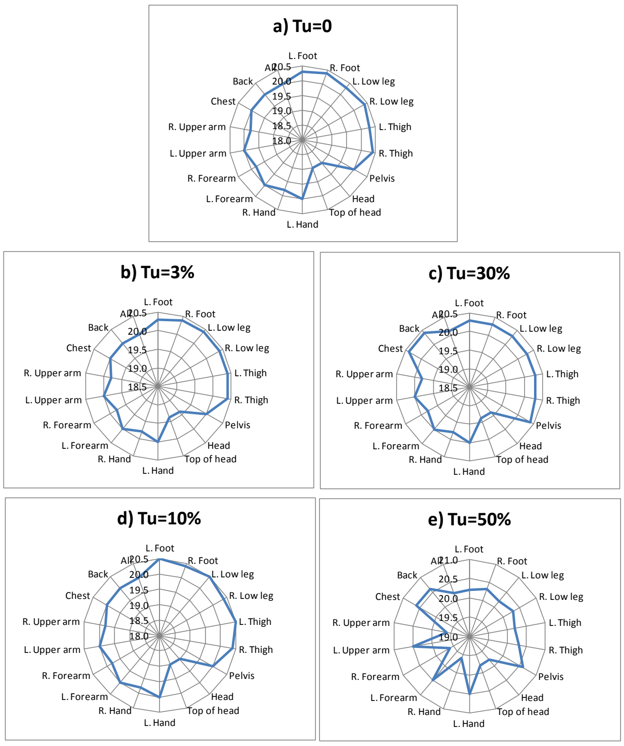

Fig.8: Distributions of the equivalent temperature t_{eq} for M1 case: a) Tu=0% b) Tu=3%, c Tu=30%, d) Tu=10%, e) Tu=50%

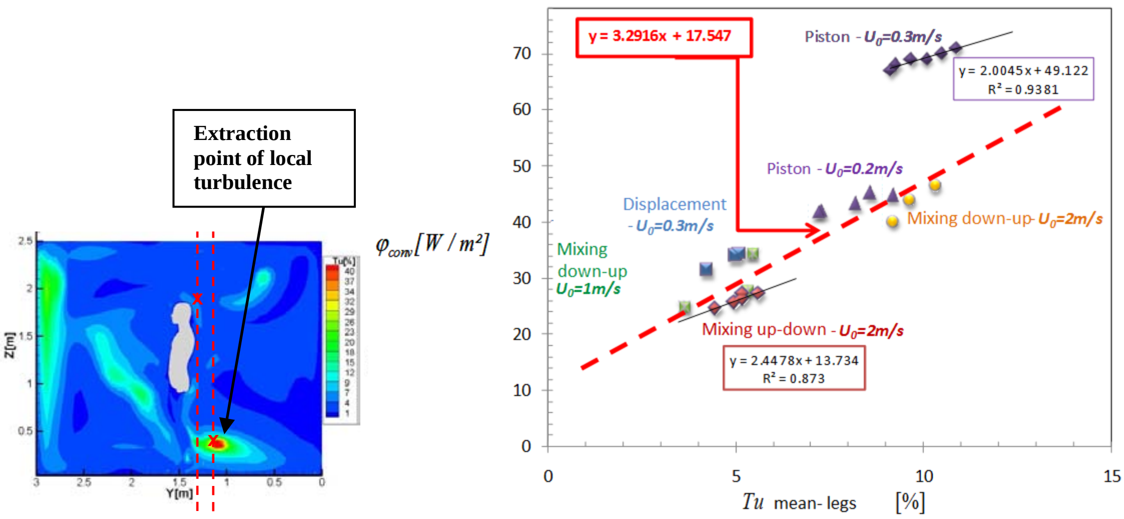

Fig. 9: Correlations between mean values of turbulence intensities and convective heat flux in the legs region for all the studied cases

CONCLUSION

The four strategies of ventilation (up-down, down-up, displacement and piston) are studied using classical thermal comfort indexes and original correlations between the degree of local turbulence and the convective heat flux exchanged between the human body and its environment. For each of these cases, we observe an intensification of convective heat transfer expressed by an average increase of 14% of convective heat flux with a maximum increase of almost 18% for the strategy of mixing ventilation with the inlet at the lower part of the room. Varying only one parameter (air turbulence intensity imposed at the inlet device) we obtain high differences on convective heat loss, fact that implies a change in thermal comfort state. This approach can determine an energy efficiency and economy in exploitation of HVAC systems. In conclusion, we can say that the analysis of convective heat flux distributions on the surface of the thermal manikin heat reveals the importance of a detailed study of the dynamics of convective flows around the body for evaluating thermal comfort. This finding is reinforced by the strong correlation between the local convective flux and local turbulence intensity.

ACKNOWLEDGEMENTS

„This work was supported by the grants of the Romanian National Authority for Scientific Research, CNCS – UEFISCDI, project numbers: PN-II-PD-PCE-2011- 3-0099, PN-II-ID-PCE-2011-3-0835 and PN-II-PT-PCCA-2011-3.2-1212".

REFERENCES

1. Abadie, M., *Contribution à l'étude de la pollution particulaire : rôle des parois, rôle de la ventilation*, in *LEPTAB*. 2000, Université de La Rochelle.
2. Meslem, A., *Contribution à l'étude du couplage thermique entre un jet et un local climatisé.* 1997, INSA Lyon.
3. Fanger, P.O., *Human requirements in future air conditioned environments* International Journal of Refrigeration, 2001. **24**: p. 148-153.
4. Fountain, M.E., *Laboratory studies of the effect of air movement on thermal comfort: a comparison and discussion of methods.* ASHRAE Transactions, 1991. **97**(1): p. 863-873.
5. Djongyang, N., R. Tchinda, and D. Njomo, *Thermal comfort : A review paper.* Renewable and Sustainable Energy REviews, 2010. **14**: p. 2626-2640.
6. Gao, N. and J. Niu, *CFD study on micro-environment around human body and personalized ventilation.* Building and Environment, 2004. **39** p. 795 - 805.
7. Cropper, P.C., et al. *Exchange of simulation data between CFD programmes and a multi-segmented human thermal comfort model*. in *Air Conditioning and the Low Carbon Cooling Challenge*. 2008. Windsor, UK.
8. Inard, C., H. Bouia, and P. Dalicieux, *Prediction of air temperature distribution in buildings with a zonal model.* Energy and Buildings, 1996. **24**(2): p. 125-132.
9. Inard, C., P. Depecker, and J.-J. Roux, *Un modèle simplifié pour la prédiction du champ de température dans les bâtiments - A simplified model for the prediction of air temperature distribution in buildings.* Révue Générale de Thermique, 1997. **36**(2): p. 113-123.
10. Inard, C., et al., *Structure moyenne et anlyse intégrale du panache thermique des convecteurs électriques.* Révue Générale de Thermique, 1997. **36**: p. 495-509.
11. Sorensen, D.N. and L.K. Voigt, *Modeling airflow and heat transfer around a seated human body by computational dynamics.* Building and Environment, 2003. **38**(6): p. 753-762.
12. Meslem, A., et al., *Numerical study a turbulent jet flow issued from a lobed orifice.* Scientific Journal of Technical University of Civil Engineering in Bucharest - Series: Mathematical Modelling in Civil Engineering, 2010. **6**(3): p. 42-51.
13. Awbi, H.B., *Ventilation of Buildings*. 2003, London, U.K.: E&FN SPON. 313.
14. Fanger, P.O., et al., *Air turbulence and sensation of draught.* Energy and Buildings, 1988. **12**(1): p. 21-39.
15. Fanger, P.O., *The new comfort equation for indoor air quality.* Ashrae Journal, 1989. **31**(10): p. 33-38.
16. Catalina, T., J. Virgone, and F. Kuznik, *Evaluation of thermal comfort using combined CFD and experimentation study in a test room equipped with a cooling ceiling.* Building and Environment, 2009. **44**(8): p. 1740-1750.
17. Zhang, Y. and R. Zhao, *Overall thermal sensation, acceptability and comfort.* Building and Environment, 2008. **43**(1): p. 44-50.
18. Zhang, Y. and R. Zhao, *Effect of local exposure on human responses.* Building and Environment, 2007. **42**(7): p. 2737-2745.