International Journal of Mechanical, Industrial and Aerospace Sciences
ISSN: 2517-9950
Vol:8, No:2, 2014

Numerical Investigation of Displacement Ventilation Effectiveness

Ramy H. Mohammed

Abstract**—**Displacement ventilation of a room with an occupant is modeled using CFD. The geometry of manikin is accurately represented in CFD model to minimize potential. Indoor zero equation turbulence model is used to simulate all cases and the effect of the thermal radiation from manikin is taken into account. After validation of the code, predicted mean vote, mean age of air, and ventilation effectiveness are used to predict the thermal comfort zones and indoor air quality. The effect of the inlet velocity and temperature on the thermal comfort and indoor air quality is investigated. The results show that the inlet velocity has great effect on the thermal comfort and indoor air quality and low inlet velocity is sufficient to establish comfortable conditions inside the room. In addition, the displacement ventilation system achieves not only thermal comfort in ventilated rooms, but also energy saving of fan power.

Keywords**—**Displacement ventilation, Energy saving, Thermal comfort, Turbulence model.

I. Introduction

DISPLACEMENT ventilation has been shown to be an effective means to remove excess heat and achieve good air quality in comparison with the conventional mixing ventilation. The technique of positive displacement has been widely used in many countries. Displacement ventilation of a room takes place by introducing the supply air at low level and at low velocity. The air enters at a temperature slightly less than that of the average room air and density difference causes the supply air to form a layer over the floor. Warm sources in the room (occupants, appliances) cause the lower level air to rise and form convective plumes that remove heat and contaminants from the sources. The warm, contaminated air is then extracted at, or near, ceiling level. The system is able to provide an environment of improved air quality as compared with the mixing of air, which occurs in conventional HVAC systems (for the same air flow rate conditions). Also, the same heat loads can be removed for a supply air temperature of typically 19^{o}C, as compared with an air temperature of about 13^{o}C in HVAC systems. As a result of thermal comfort limitations given in BS EN ISO Standard 7730 [1], and currently assumed by practitioners to hold well in displacement ventilation environments (namely, that the vertical air temperature difference, VATD, should be less than 3K m^{-1}), a displacement ventilation system is limited to removing a convective load of up to about 30 W m^{-2} of floor area.

R. H. Mohammed is with the Zagazig University, CO 44519 Egypt, (phone: 201114049331; e-mail: Rhamdy@ zu.edu.eg).

The effect of people on the surrounding airflow can be significant and hence should be considered when assessing thermal comfort. Computational fluid dynamics (CFD) is a useful tool in studying ventilation both for assessment of an existing configuration and in the design process.

CFD studies can often be cheaper than experiments and they provide a wealth of data on the details of the flow in the room, which can be useful for flow visualization.

An early application of CFD to this area was made by Murakami et al. [2] who studied the convective heat transfer between a model of an unclothed manikin and its environment. Five different scenarios for flow around a standing person were examined, including a number of uniform flows past the person and a displacement ventilation scenario.

Nilsson and Holmer [3] modeled the flow around a seated person in a room with displacement ventilation using the RNG k-ε turbulence model [4]. The results were converted into thermal comfort indicators and compared with measurements. Murakami et al. [5] modeled the flow around a standing person in a room with displacement ventilation. A low-Reynolds number k-ε turbulence model was used and thermal radiation was modeled using the method of Gebhart [6]. Murakami [7] made an in-depth study of the flow around a standing person in a room with displacement ventilation, emphasizing the importance of modeling thermal radiation in such flows. The influence of breathing on the thermal plume was considered using both thermal manikins and real people. A low-Reynolds number k-ε turbulence model was used for the majority of the analysis of the mean flow around the person. In addition, large-eddy simulation (LES) was used to study the flow around a simplified rectangle used to approximate the standing person. However, results were only presented of velocity and temperature power spectra at specific locations.

A few studies have been made of convection heat transfer conditions in rooms with displacement ventilation and chilled ceilings. The work reported by Novoselac et al. [8] was mostly concerned with floor heat transfer in rooms with displacement ventilation alone the floor gravity current being rather different to that in other forms of mechanical ventilation. The studies by Jeong et al. [9] examined existing correlations for ceilings and the study by Karadag [10] was based on numerical convection heat transfer results and theoretical radiant heat transfer models rather than experimental measurements. Andrés-Chicote et al. made an experimental study of convection from a chilled ceiling in a full size test room but without any plume heat sources. Causone et al. [11]

also carried out full scale experiments in an unventilated test chamber. They comment on some systematic differences in measured ceiling convection coefficients when compared to other work and suggest this may be due to different reference temperature definitions but they also point out some systematic differences between cases with heated walls as the heat source and cases with plume heat sources. The physical modeling study by Thomas et al. [12], which employed an unventilated heated water tank with a small plume, did not quantify convection heat transfer coefficients but did highlight the existence of Rayleigh-Bérnard type convection cells close to the ceiling surface under steady state conditions. These cells seem to have been confined to a relatively shallow layer next to the ceiling surface and not induced large scale mixing in the upper part of the room. The authors suggest further experiments to investigate to what extent the Rayleigh-Bérnard cells at the surface would be influenced by other features or turbulent structures in the flow.

Sideroff and Dang [13] carried out simulations of the displacement ventilation test case proposed as part of the benchmark exercise. The standard *k-ε* and v^2f turbulence models were used, and the v^2f model [14] was found to return better results in the thermal plume. However, a thermal radiation model was not used a factor that may have a significant influence on the flow and hence the assessment of the predictions from different turbulence models. Recent work on the same test case [15] presented steady state results using a *k-ε* turbulence model and a thermal radiation model. A simplified geometry made from four rectangles was used to represent the manikin head, torso and legs. In general, the results showed good agreement with the experimental data and the authors concluded that accurate boundary conditions for the body were more important than geometrical accuracy.

From the view point of HVAC design as present in ASHRAE Applications Handbook [16] that recommends general guidelines for an operating room that its temperature should be held in the range of (20-24^{o}C), relative humidity should be from 50% to 60%, positive air pressure should be maintained, and all air exhausted with no recirculation is preferred. ISO7730-1994 presents moderate thermal environments and recommends that the vertical air temperature difference (VATD) from the ankle level 0.1m to the height of 1.1m which is representative for the breathing zone of a sitting person has been used to determine the local thermal discomfort. The maximum temperature difference between ankle level and head one must be below 3^{o}C [17].The previous work emphasized on finding the suitable turbulence models for simulating ventilated rooms. Also, the effect of person on the airflow characteristics and the radiation model had been neglected in many studies. The information related to the relationship among displacement ventilation, thermal comfort, indoor air quality, and energy consumption is not enough.

The important aim of this study is to improve the analysis of real life environments. Thus, particular emphasis is placed on the complication of human shape and investigates its effect on the indoor air quality and thermal comfort. Also, it tries to study the effectiveness of the displacement ventilation and its impact on the energy consumption and thermal comfort.

II. Numerical CFD Model

A. Airflow Model

A commercial CFD code [*AirPak 2.0.6*] is used to predict flow and temperature field in the room with displacement ventilation system. The AirPak program is a customized version of the FLUENT program [18]. The governing equations are steady state, incompressible Navier-Stokes equations, and indoor zero-equation turbulence model in Cartesian coordinates [19]. Indoor airflow calculations use the Boussinesq approximation for thermal buoyancy [20]. The approximation takes air density as constant in the momentum terms and considers the buoyancy influence on air movement by the difference between the local air weight and the pressure gradient. With an eddy-viscosity model, the indoor airflow is described by the following time-averaged Navier-Stokes equations for the conservation of mass, momentum, energy, and species transport equation. Also, discrete ordinates radiation model has been used to simulate the radiation effect.

- **Continuity**

$$\frac{\partial V_i}{\partial x_i}=0 \qquad (1)$$

where V_i= mean velocity component in xi-direction; x_i = coordinate (for i =1, 2, 3, x_i corresponds to three perpendicular axes).

- **Momentum Equation**

$$\frac{\partial \rho V_i}{\partial t}+\frac{\partial \rho V_i V_j}{\partial x_j}=-\frac{\partial P}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu_{eff}\left(\frac{\partial V_i}{\partial x_j}+\frac{\partial V_j}{\partial x_i}\right)\right]+\rho\beta(t_o-t)g_i \qquad (2)$$

where ρ= air density; V_j = velocity component in x_j-direction; P = pressure; μ_{eff} = effective dynamic viscosity; β= thermal expansion coefficient of air; t_o = temperature of a reference point; t= temperature; g_i= gravity acceleration in i-direction. The last term on the right side of (2) is the buoyancy term. The turbulent influences are lumped into the effective viscosity as the sum of the turbulent viscosity μ_t and laminar viscosity μ_l:

$$\mu_{eff}= \mu_t + \mu_l \qquad (3)$$

In the Prandtl-Kolmogorov assumption, the turbulent viscosity is the product of turbulence kinetic energy, k, and turbulent macro scale, L, that is a proper length scale for turbulence interactions: $\mu_t = C_v \rho k^{1/2} L$ where C_v = 0.5478, an empirical constant.

- **Energy Equation**

$$\frac{\partial \rho t}{\partial t}+\frac{\partial \rho V_j t}{\partial x_j}=\frac{\partial}{\partial x_j}\left[\Gamma_{T,eff}\left(\frac{\partial t}{\partial x_j}\right)\right] \qquad (4)$$

where Γ_{eff} is effective turbulent diffusion coefficient for t, $\Gamma_{T,eff}=\mu_{eff}/Pr_{eff}$ where Pr_{eff} is effective Prandtl number.

- **The concentration of species equation**

$$\frac{\partial \rho C}{\partial t}+\frac{\partial \rho V_j C}{\partial x_j}=\frac{\partial}{\partial x_j}\left[\Gamma_{c,eff}\left(\frac{\partial C}{\partial x_j}\right)\right]+Sc \qquad (5)$$

where C is species concentration; $\Gamma_{c,eff}$ is effective turbulent diffusion coefficient for C; Sc is source term of C. Similar method to the energy equation is used to determine the effective diffusive coefficient for species concentration $\Gamma_{C,eff}=\mu_{eff}/Sc_{eff}$ where effective Schmidt number, Sc_{eff}, is equal to 1.0.

B. Thermal Sensation

Thermal sensation is evaluated in terms of the predicted mean vote (PMV) and the predicted percentage of dissatisfied (PPD) proposed by Fanger [21]. PMV is a function of air velocity, air temperature, mean radiant temperature, water vapor pressure of air, clothing thermal resistance and occupant's metabolic rate. PPD (%) is related to PMV by the following equation [22]:

$$PPD = 100 - 95 \exp - (0.03353PMV^4+0.2179PMV^2) \qquad (6)$$

The mean age of air (M) is generally defined as the average time for air to travel from a supply inlet area to any location in a ventilated room [23], [24]. The mean age of air is one of the most important parameters that describe the ventilation efficiency in a space. The age of air concept assumes that the age of air at the inlet area is equal to zero (100% fresh). It is obvious that the high values of the mean age of air (M) mean that part of the air circulates for a long time inside the room. So the values of the mean age of air reflect the efficiency of ventilation system. It can be calculated according to ASHRAE standard from the following transport equation:

$$\frac{\partial}{\partial x_i}\left(\rho u_i M-\left(2.88\rho\times10^{-5}+\frac{\mu_{eff}}{0.7}\right)\frac{\partial M}{\partial x_i}\right)=\rho \qquad (7)$$

The overall ventilation effectiveness (E) represents the effectiveness of energy utilization supplied into the occupied zone to achieve a satisfactory thermal comfort level [23], [24]:

$$E=\frac{t_e-t_s}{t_m-t_s}x100 \qquad (8)$$

C. Boundary Conditions

The size of room is 3.5x3x2.5 m and the person is located at its center as shown in Fig. 1. It has inlet at the floor level and outlet at the ceiling level. No-slip boundary conditions are used along the room walls, the ceiling, and the floor. All walls and ceiling are isothermal with specified temperature. The floor is assumed to be adiabatic wall which the temperature gradient or heat transfer flux is zero. For all cases, 8% turbulence intensity is used. At the outflow boundary, environmental pressure is considered where the boundary values are estimated by extrapolation from the interior points. The present CFD code required initial conditions throughout the domain to start the solution. The boundary conditions are shown in Table I.

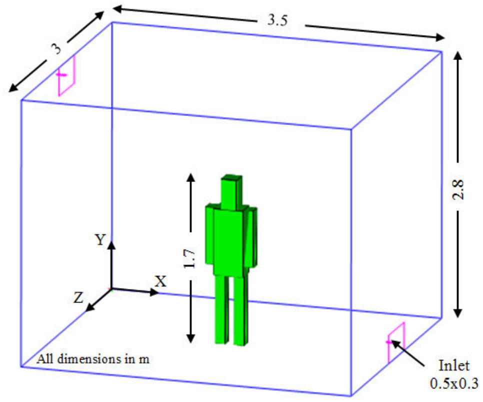

Fig. 1 Computational domain

TABLE I
BOUNDARY CONDITIONS

Boundary	Velocity (m/s)	Temperature (°C)	Relative Humidity (%)
Supply	−Vx = 0.1- 0.4, Vy = Vz = 0.0	18-24	60
Exhaust	(Vx, Vy, Vz)[a]	(t)[a]	(RH)[a]
Walls and ceiling	Vx = Vy = Vz = 0.0	28	0
Floor	Vx = Vy = Vz = 0.0	Adiabatic (q = 0)	0
Person		Heat generation = 1.0 met and clothes insulation = 1.0 clo.	

[a] Unknown to be solved as a part of numerical solution.

D. Mesh Sensitivity

Different structured grids are used to establish the grid independence. Fig. 2 shows the vertical temperature distribution averaged over the horizontal plane for 200,000, 400,000, 600,000 and 800,000 grid sizes. In this graph, the results 600,000 and 800,000 are very close to one another and therefore the results obtained with 800,000 grid size can be considered to be grid independent.

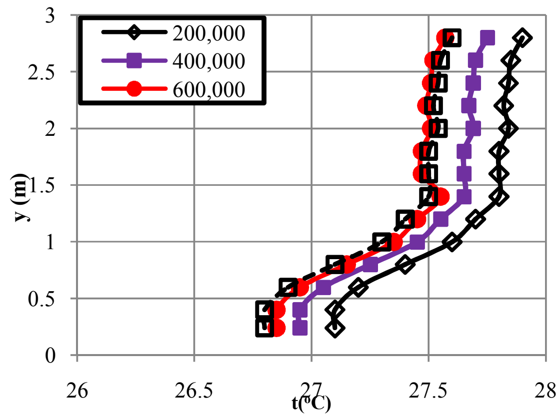

Fig. 2 Temperature distribution over vertical line at the center of the room for different grid sizes

E. Code Validation

The accuracy of the CFD code is established through a validation process. This process involves comparison of predicted results from the CFD code with those obtained from pervious numerical studies with same conditions. One case has been chosen from present study and compared with the results taken from Spitler's experimental work [25], Baker's numerical study [26], and Alfahaid's numerical study [27] with the same parameters. Fig. 3 shows the temperature distribution on vertical line at the center of the room. Accordingly, indoor zero equation turbulence model can be used to simulate indoor airflow with good agreement and accuracy of nearly 0.88%.

III. Results and Discussion

A. Description of Airflow Emerged from the Inlet

The air flow emerged from the opening enters the room in one direction and suddenly expands to fill the whole room as shown in Fig. 4. The cone of the airflow vanishes when its front reaches to the person's legs therefore; presence of human increases the turbulence and disturbance in the flow distribution.

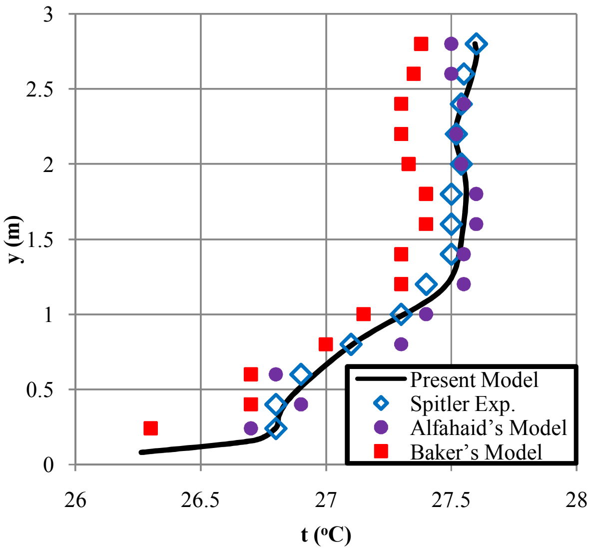

Fig. 3 Comparison between the present model and previous experimental and numerical data

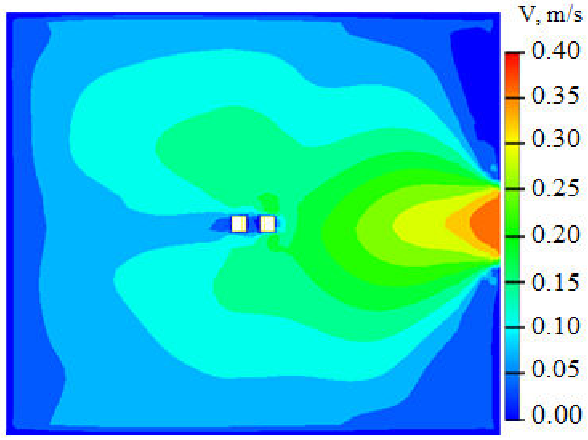

Fig. 4 Distribution of air velocity at y=0.15m

B. Effect of Inlet Velocity

Different inlet velocities have been used to study the impact of inlet velocity on thermal and flow fields. Fig. 5 indicates the velocity distribution at vertical plane at the center of the room. The velocity distribution is nearly uniform entire the room except above the person's head, but it is still in the comfort zone at 0.2 m/s for all cases. The effect of the buoyancy force is clear and significant at low inlet velocity and vanishes when inlet velocity increases. The temperature distribution inside the room is uniform at low inlet velocity and its stratification at higher velocity is well established as indicated in Fig. 6 and the maximum value occurs at the surface of person.

The air circulates for long time in the upper half of the room especially at the corners as illustrated in Fig. 7. As the inlet velocity increases, the mean age of the air in the room decreases. In spite of change of the inlet velocity, the distribution of mean age of the air remains constant and its stratification is well established.

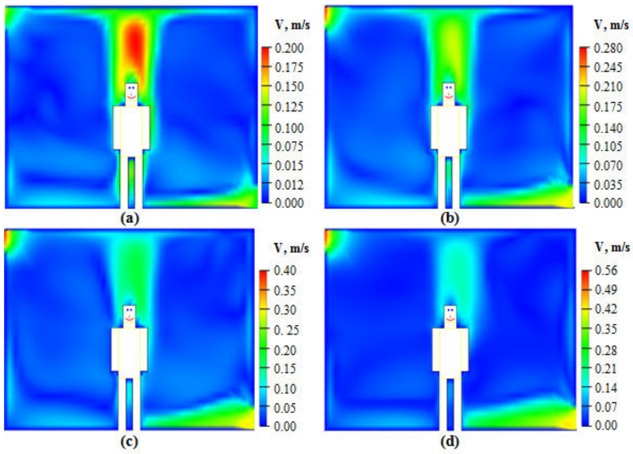

Fig. 5 Velocity distribution on vertical plane at inlet velocity of (a) 0.1m/s, (b) 0.2m/s, (c) 0.3m/s, and (d) 0.4m/s

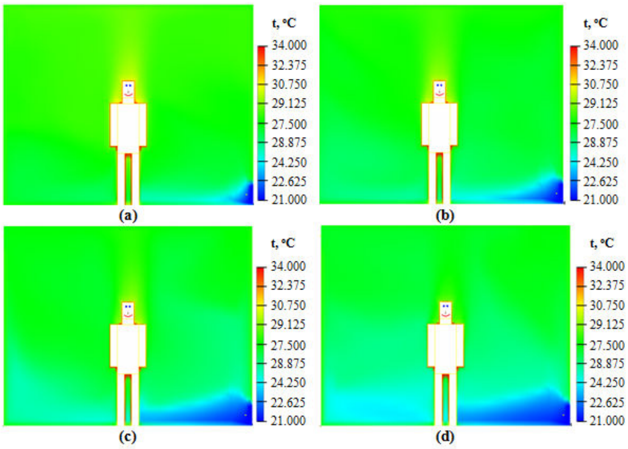

Fig. 6 Temperature distribution on vertical plane at inlet velocity of (a) 0.1m/s, (b) 0.2m/s, (c) 0.3m/s, and (d) 0.4m/s

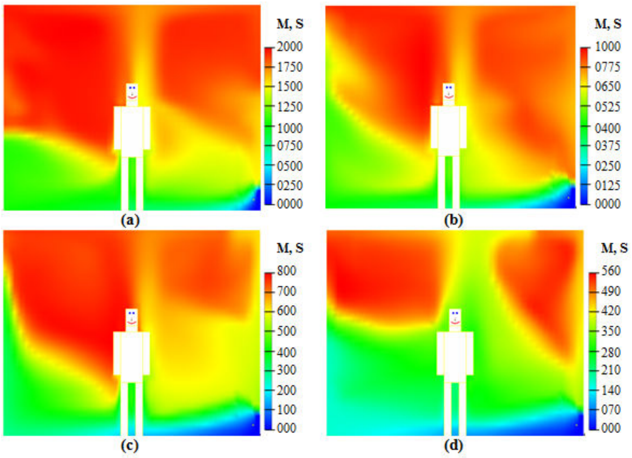

Fig.7 Distribution of mean age of the air at inlet velocity of (a) 0.1m/s, (b) 0.2m/s, (c) 0.3m/s, and (d) 0.4m/s

Fig. 8 indicates the change of PMV, vertical air temperature difference, and ventilation effectiveness at different Reynolds number. It is clear that the effectiveness is nearly constant as inlet velocity increases. On the other hand, PMV declines and VATD rises when inlet velocity increases. In spite of these variations in these parameters, the conditions of indoor air quality and thermal comfort have been achieved at all cases and room seems to be cooler at high inlet velocity. Also, low inlet velocity is enough to satisfy ISO7730 recommendations. As a result, the increase of the inlet velocity is not preferable, because it will raise the operating energy without any influence and may increase VATD above 3 and this is not recommended.

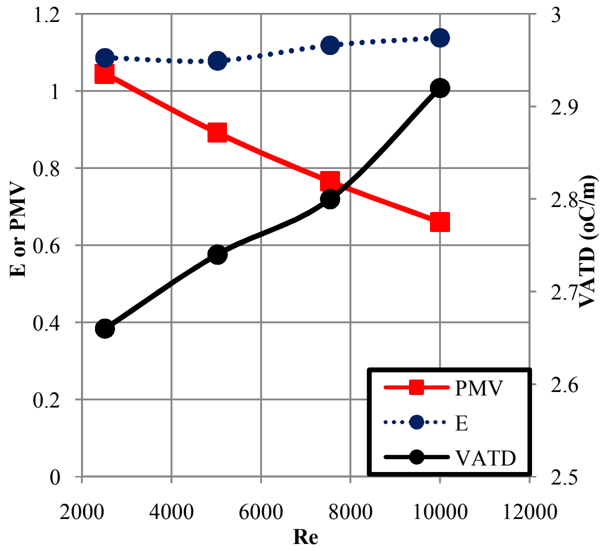

Fig. 8 Vertical air temperature difference, mean PMV, and effectiveness at various Reynolds numbers

C. Effect of Inlet Temperature

The impact of the inlet temperature on the distribution of velocity and mean age of the air inside the room can be neglected as shown in Fig. 9 and hence it has not any effect on the room air flow distribution. Fig. 10 shows the variation of PMV, E, and VATD at different inlet temperature at lower inlet velocity. It is clear that the VATD noticeably declines and PMV steadily increases when inlet temperature increases, but the ventilation effectiveness remains nearly constant and it does not depend on the inlet temperature. The room seems to be warmer at high inlet temperature. Similarly, the trend of PPD is similar to the trend of the PMV.

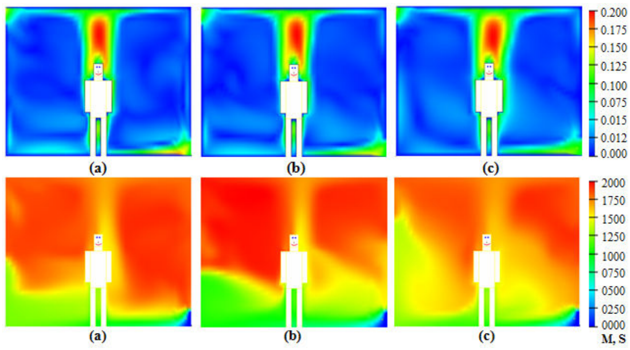

Fig. 9 Distribution of velocity and mean age of the air at inlet temperature of (a) 18, (b) 21, and (c) 23^{o}C

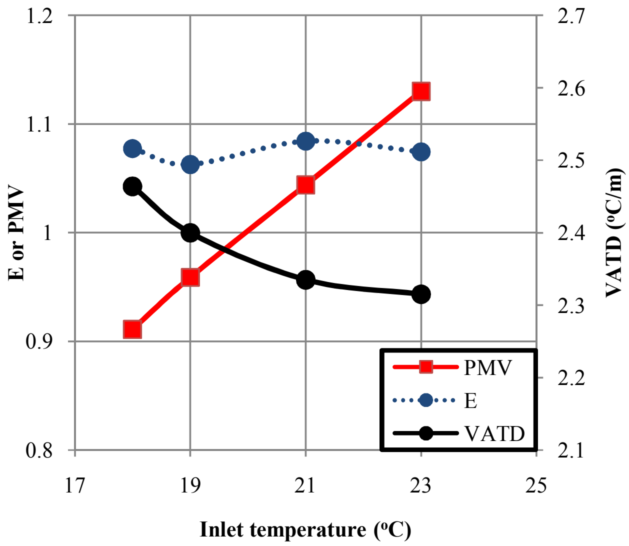

Fig. 10 Effect of the inlet temperature on PMV, E, and VATD

IV. Conclusions

Numerical investigation to study the effectiveness of the displacement ventilation has been reported for several inlet conditions. The effect of inlet velocity and temperature on the room air flow characteristics, indoor air quality, and thermal comfort has been demonstrated. CFD, an important tool in HVAC engineering, is used to analyze the data generated by models representing the complexity of the flow patterns that evolve inside the ventilated spaces. The indoor zero equation, which is more simple, can be used to simulate the airflow in case of the displacement ventilation with good accuracy.

The effect of the inlet velocity on the thermal comfort and indoor air quality is higher than that of the inlet temperature and lower inlet velocity is sufficient to establish thermal comfort zone inside the rooms and it is enough to satisfy ISO7730 recommendations. So, we should know that the increase of the inlet velocity, in the permissible range, has not any effect of the thermal comfort, but it may be effective if removing more contaminates from the room is demanded and required. Also, the findings indicate that there is a conflict between VATD and PMV and low velocity is suitable to achieve both conditions.

Finally, displacement ventilation system can achieve not only thermal comfort in ventilated rooms, but also energy saving of fan power.

References

[1] ISO, BS EN ISO Standard 7730. Moderate thermal environments determination of the PMV and PPD indices and specification of the conditions for thermal comfort, International Standards Organization (1995).
[2] S. Murakami, S. Kato and J. Zeng, Flow and temperature fields around a human body with various room air distributions: CFD study on computational thermal manikin—part I, ASHRAE Transactions 103 (1997) 3-15.
[3] H. O. Nilsson, I. Holmer, Comfort climate evaluation with thermal manikin methods and computer simulation models, Indoor Air 13 (2003) 28-37.
[4] V. Yakhot, S. A. Orszag, Renormalization group analysis of turbulence. I. Basic theory, Journal of Scientific Computing 1 (1986) 3-51.
[5] S. Murakami, S. Kato, J. Zeng, Combined simulation of airflow, radiation and moisture transport for heat release from a human body, Building and Environment 35 (2000) 489-500.
[6] B. Gebhart, A new method for calculating radiative exchanges, ASHRAE Transactions 65 (1959) 321-323.
[7] S. Murakami, Analysis and design of the micro-climate around the human body with respiration by CFD, Indoor Air 14 (2004) 144-156.
[8] A. Novoselac, B.J. Burley and J. Srebric, Development of new and validation of existing convection correlations for rooms with displacement ventilation systems, Energy Build 2006; 38(3):163-73.
[9] J.W. Jeong, S. A. Mumma, Ceiling radiant cooling panel capacity enhanced by mixed convection in mechanically ventilated spaces, Appl Thermal Eng 2003; 23(18) 2293-306.
[10] R. Karadag, The investigation of relation between radiative and convective heat transfer coefficients at the ceiling in a cooled ceiling room, Energy Conversion Manag 2009; 50(1) 1-5.
[11] F. Causone, S.P. Corgnati, M. Filippi, B. W. Olesen, Experimental evaluation of heat transfer coefficients between radiant ceiling and room, Energy Build 2009; 41(6) 622-8.
[12] L. P. Thomas, B.M. Marino, R. Tovar, J. Castillo, Flow generated by a thermal plume in a cooled-ceiling system, Energy Build 43(10) (2011) 2727-36.
[13] C. N. Sideroff, T. Q. Dang, Challenges in evaluating turbulence models with benchmark cases, in: ASHRAE Summer Meeting, Denver, Colorado, 2005.
[14] P.A. Durbin, Near-wall turbulent closure modelling without 'damping functions', Theoretical and Computational Fluid Dynamics 3 (1991) 1-13.
[15] J. Srebric, V. Vukovic, Simplified physical and simulation modeling of building occupants a seminar, in: Experimental and CFD Benchmark Studies of Indoor Flow around Thermal Manikins". ASHRAE winter meeting, Chicago, 2006.
[16] ASHRAE Handbook: Applications, American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc., Atlanta, GA, 2003.
[17] ISO.ISO7730: moderate thermal environments determination of the PMV and PPD indices and specification of the conditions for thermal comfort. Geneva: International Organization for Standardization, 1994.
[18] Fluent Inc. 2001 Getting Started with Airpak 2.
[19] Q. Chen, W. Xu, A zero-equation turbulence model for indoor airflow simulation, Energy and Buildings 28 (1998) 137-144.
[20] D. Etheridge, M. Sandberg, Building ventilation, theory and measurement, England: John Wiley and Sons Ltd., 1996.
[21] P. O. Fanger, Thermal Comfort- Analysis and Applications in Environmental Engineering, Robert E. Krieger, Florida, 1982.
[22] Moderate thermal environments -- determination of PMV and PPD indices and specification of the conditions for thermal comfort, ISO 7730, International Standards Organization, Geneva, Switzerland, 1984.
[23] M. Sandberg, M. Sjöberg, The use of moments for assessing air quality in ventilated rooms. Building and Environment 18 (1983) 181-197.
[24] M. A. Aziz, Ibrahim A.M. Gad, E.F.A. Mohammed, R. H. Mohammed, Experimental and numerical study of influence of air ceiling diffusers on room airflow characteristics, Energy and Buildings 55 (2012) 738-746.
[25] J. D. Spitler, An experimental investigation of air flow and convective heat transfer in enclosures having large ventilative flow rates, Ph.D. Thesis, University of Illinois at Urbana-Champaign, 1990.
[26] A. J. Baker, P. T. Williams, and R. M. Kelso, Numerical Calculation of Room Air Motion - part 1: Math, Physics and CFD Modeling, ASHRAE Transactions 100 Part1 514-530.
[27] A. F. Alfahaid, Effects of ventilation on human Thermal comfort in rooms, Ph.D. Thesis, Old Dominion University, December 2000.