Energy Saving by Using a Local Ceiling Spot Air Distribution System in an Equipped Office Room

Wisam A M Al-Shohani *, Ahmed Qasim Ahmed*, Ahmed Jawad Khaleel*, Hassan J Dakkama‡, Hayder Obaida*, Tareq Hamid Fayyad**

* Engineering Technical College, Middle Technical University, Baghdad, Iraq

** Economic Directorate, Ministry of Oil, Baghdad, Iraq

(wabd1984@yahoo.com, en_ahmed82@yahoo.com, ahmed1982_jk@yahoo.com)

‡ Corresponding Author; Hassan J Dakkama, Engineering Technical College, Middle Technical University, Baghdad, Iraq, hassan_jfsd@yahoo.com

Received: 17.07.2019 Accepted:23.09.2019

Abstract- In this paper, a proposed Ceiling Personalized Ventilation (CPV) system was studied numerically to explain the influence of the proposed system on both of energy demand and thermal human comfort. This system was installed on the ceiling above the occupant zone to make sure that the fresh air supplied directly to the occupant zone. In addition, this system considers the main ventilation system in the investigated room and targeted the occupied zone only with limited working hours. A validated CFD model was used in this examination. The results show that an insulated healthy zone will appear and bounded the area around the occupant by using the proposed CPV system. This zone will enhance the conditions of the occupant environment and protect the working area from the unwanted thermal effects. Also, the results show that using this system will provide a supplied fresh air directly to the occupied area and this will reduce the demand on energy by reducing the load on the cooling coil. Therefore, using this system will improve the local thermal environment and saving more energy.

Keywords- Energy saving, Indoor Thermal comfort, Spot cooling, CFD, Ventilation

Nomenclature

Abbreviations

DV	displacement ventilation
HVAC	heating, ventilation and air conditioning system
IAQ	indoor air quality
PV	Personalised Ventilation
LEV	Local Exhaust Ventilation
MV	Mixing ventilation
CPV	Ceiling Personalized Ventilation

Greek letters

β	coefficient of thermal expansion (1/K)
ε	turbulent dissipation rate (m^2/s^3)
μ	dynamic viscosity (kg/(m s)
ρ	fluid density (kg/m^3)
σ_k	model constant for k equation of the turbulence model
C_μ	model constant of the turbulence model
c_p	specific heat of air (J/(kg k)
S	mean strain rate tensor magnitude
S_{ij}	strain rate tensor
R_k	additional term in the turbulence model
i, j	trajectories

1. Introduction

Humans around the world spend about 90% of their time indoors, this includes the rest and work times [1]. Depending on the information from the development indicators for the world, the density of the population will increase dramatically in the next 30 years. 70% of their increase will be in urban centers. This will lead to an increase in the buildings number. About 20 - 40 % of the total energy are consumed by buildings sector [2]. Half of this per cent goes to operate Heating, Ventilation and Air-Conditioning (HVAC) systems to maintain and create a healthy and suitable indoor conditions [3]. This per cent is too high and needs to be minimized without effect on indoor thermal comfort. The air temperature, velocity, relative humidity and indoor air quality (IAQ) for the indoor play a central role in generating a healthy and suitable working area [4, 5]. The efficient distribution of the indoor air is influence positively on thermal comfort and the amount of energy demand [6-10]. For this reason, efficient design of an air distribution system will save more energy and this can be achieved by using different techniques to minimize the waste of treated air [11, 12]. Many techniques are employed to generate a comfortable local area at low energy consumption. One of these effective techniques is Personal Ventilation (PV).This method used to create a suitable local area around the occupants at low energy consumption. Also, this system can provide fresh air directly toward the personal area. In addition, this method has the ability to control the supplied air temperature of the targeted area as well as, the ability to control the air direction flow and its mass flow rate [13, 14]. The evaluation of the indoor environment needs a careful balance between thermal human comfort and demand on energy. It is very important to evaluate these two parameters, energy demand and thermal human comfort, at the same time [15-17]. The impact of the indoor air parameters distribution on thermal comfort and energy saving was investigated by Frontczak and Wargocki [16]. They found that the efficient distribution of the indoor air will provide good thermal comfort and efficient energy saving. Another study that investigated the local PV system was performed by Melikov [18] that studied the effect of using a PV system on human thermal comfort and energy consumption as well as human response to this type of ventilation system. The results showed that various parameters which may affect the on human comfort and energy saving such as flow rate, flow direction and supplied temperature. The concept of using local ventilation was examined to show the ability of using this system to enhance the supplied air quality at the inhaled zone. This study was performed by Halvonova et al. [19]. In this study, different amounts of air flow rate and temperatures value were tested. They found that using a suitable amount of air flow and accurate supplied air temperature will help to provide a good indoor environment. Experimental study was performed by Taheri et al. [20] to evaluate the air quality for the indoor and thermal comfort in an equipped office room. In their investigation, the displacement ventilation (DV) system, PV system and natural ventilation system were employed and tested for each case study. The results show that a good indoor environment with an acceptable amount of energy consumption was achieved when using the PV system comparing with the DV system and natural ventilation system. Another experimental study was conducted by Sekhar et al. [21] to examine the influence of using the PV system on indoor thermal conditions and its ability to reduce the energy consumption under a humid and hot climate in Singapore. The results showed that the energy consumption decreases by 15-30% when using the PV systems conjunction with another ventilation background. Also, this system has the ability to enhance the air quality in breathing zones of occupants by up to 50 %. Makhoul et al. [22] performed an experimental field study to investigate the influence of using a ceiling PV system on energy demand and the indoor thermal conditions. They found that using this system was to improve the energy saving by up to 30 %. This study was improved by Makhoul et al. [23] they added desk fans inside proposed space this technique saving more energy and provide an acceptable thermal environment. A study by Cruceanu et al. [24] revealed that 60 % of energy consumption was achieved when using PV system combined with MV system comparing with using MV system alone. Another study was performed by Kaczmarczyk et al. [25] to examined 5 different air terminals device of PV in an office space. The thermal environment was evaluated for all five terminals. The results revealed that these systems enhanced the thermal conditions for the investigated room and achieved the human comfort requirements. A numerical and experimental study was performed by Makhoul et al. [22] to examine the impact of using the ceiling PV system on a local indoor environment. The results revealed that the suitable thermal comfort, good air quality and reducing energy consumption are the most important features of the ceiling PV system. Another study presented by Yang et al. [26] to ventilate a limited zone by using an air terminal device (ATD) in the ceiling as a personal ventilation system. The results showed that by using this technique the energy saving will reach 15.44% with personal ventilation temperature of 23.5oC. An experimental study by Pantelic et al. [27] showed that the PV system has a significant role in reducing the contaminant concentration and protecting the occupant from the airborne particle emitted from infected people. An investigation was performed by Bolashikov et al. [28] proposed an advanced method of air distribution using PV. Through it can significantly reduce the infection due to the low flow rates and easy of orientation to different places because of the high flexibility.

Above previous study deal with the PV system as a secondary system that coupled with the other main ventilation system such as the DV system, MV system and other ventilation systems. Inadequate studies were used in the PV system as the main ventilation system in a room. Also limited study were used this type of ventilation to target the occupied zone only for the limited working hours. Therefore, in this study, the proposed CPV system was used alone without coupled with other ventilation system to study the effect of using this system on providing a comfortable local thermal environment and saving energy. Also, this study targeted the occupied zone only in the office room with a known number of occupants and limited working hours.

2. Numerical method

2.2 Case study explanation

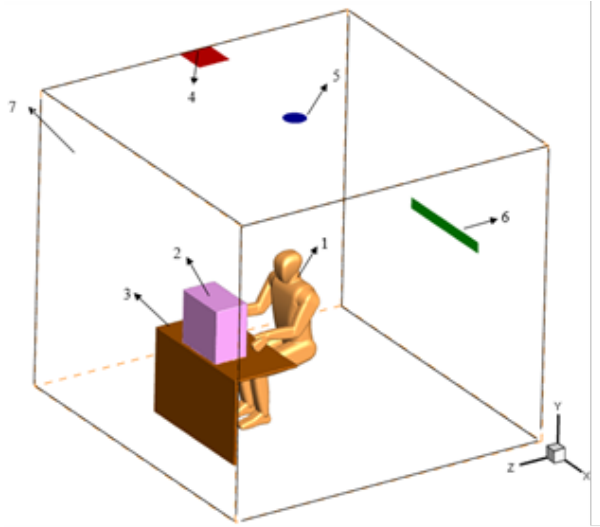
A typical equipped office room with limited working hours was used in this investigation. The dimensions of the simulated room were 2 m long, 3 m high and 3 m wide. As shown in figure 1, the heat sources included one occupant located in the room center in front of the desk with 60 W of heat flux. Another internal heat source was a computer located on the desk with 100 W of heat flux. These tow heat sources were employed in this simulation to represent the main heat sources in the occupied zone for the small office room. All side walls, ceiling, and roof were assumed to be insulated. The CPV system with 0.3 m in diameter was mounted in the ceiling room center to supply the treated air directly towards the occupied zone (see figure 1). The supply conditions of the treated air were 24 º C in temperature. The exhaust opening (0.22 m × 0.11 m) positioned at the ceiling level to extract the contaminated warm air. While the return opening with the dimension of (1m×0.1 m) located at 1.3 m from the floor at the side wall (see figure 1). In this study two different amount of the supply air which presented in Table 1 are examined to show the influence of the supply velocity of the proposed system on indoor thermal environment and energy consumption. The summarized the detailed information of the simulated room was presented in Table 2.

Table 1. Examined case study

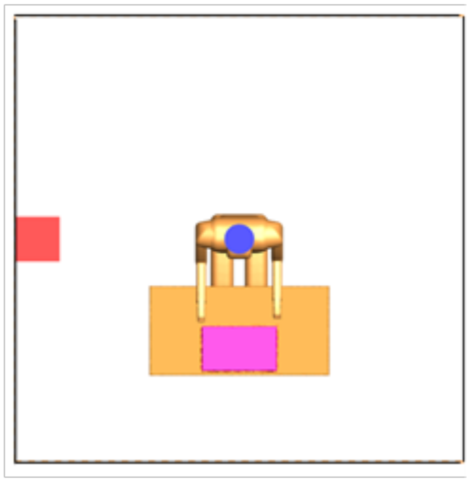
Case study	Supply velocity m/sec
Case-1	0.5
Case-2	0.7

Table .2 Detailed information of the simulated room

Room dimension	2m × 3m×3m
The dimensions of the CPV supply, Exhaust opening and return opening	0.3 m(diameter), (0.22m×0.11m) and (1m×0.1 m) respectively.
Bounded walls, ceiling, roof	insulated (No heat flux)
Heat flux of the occupant and computer	60 W and 100 W respectively
Exhaust	Pressure outlet
Supply velocity	0.3 m/sec



(a)



(b)

Fig. 1. (a) Schematic diagram of the simulated room; 1- Occupant; 2- Computer; 3-Desk; 4-Exhaust opening; 5- The CPV system; 6- Return opening 7- Adiabatic bounded walls; (b)- Top view of the simulated room.

2.2 CFD Method

2.2.1 Grid Design

The ANSYS ICEM software was employed in this investigation to design and create the grid of the simulated small room. As a complex simulated room geometry (see figure 1), a tetrahedral unstructured mesh was selected for the mesh design. This was usually used for such complex geometry. As shown in figure 2, the generated mesh was distributed carefully to cover all interested zone such as the area near occupants, computer and supplied air. For the accurate simulation results, a required y+, 3<y+<11, was taken into account in this simulation. Also, for accuracy improvement, the mesh independent test was performed. Three different size of mesh were employed in this investigation. A mesh-1, mesh-2 and mesh-3 with size of 2225000, 1720000 and 1457000 respectively (see Table 3).

The grid was clustered in an important area such as area with a high gradients of heat flow and air movement, such as area near the supply opening, internal heat sources (occupant and computer) as well as area around the working space. In order to control the total size of grid, only the size of the mesh in the investigated domain was changed without changing the surface mesh size for the occupant and computer. For this simulation, a 172000 was selected as a suitable size of the mesh. This was because that by increasing mesh size, there was no big difference in temperature and velocity. For this reason this mesh type, mesh-2, was selected for the rest of simulation.

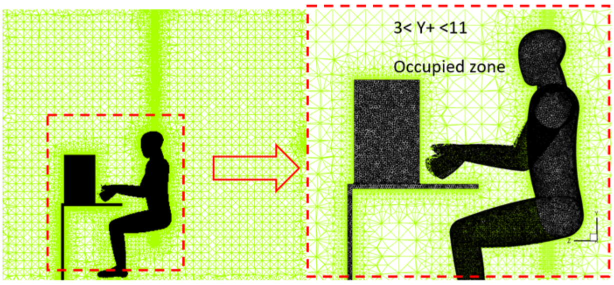


Fig. 2. Mesh design for the simulated room

Table .3 Mesh test

Mesh Type	Mesh size
mesh-1	2225000
mesh-2	1720000
mesh-3	1457000

2.1.2 Room air flow modelling

The selected turbulence model for this simulation was the (RNG) k-ε. In this model the air movement and temperature distribution are predicted correctly and accurately [29-32]. For these reasons, this model was selected to be the turbulence model for this simulation. All equation related to this model can be expressed as follow [33]:

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho k u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+P_k+\rho\varepsilon \quad (1)$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial(\rho\varepsilon u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}P_k+C_{2\varepsilon}^{*}\rho\frac{\varepsilon^2}{k} \quad (2)$$

where

$$C_{2\varepsilon}^{*}=C_{2\varepsilon}+\left(C_\mu\eta^3(1-\eta/\eta o)\right)/1+\beta\eta^3$$

with $\eta=(Sk/\varepsilon)$ and $S=\sqrt{2S_{ij}S_{ij}}$,

and

$C_\mu=0.0845$, $\sigma_k=0.7194$, $\sigma_\varepsilon=0.7194$, $C_{\varepsilon 1}=1.42$, $C_{\varepsilon 2}=1.68$, $\eta o=4.38$ and $\beta=0.012$

In this examination, the motion of the indoor air and the internal heat flow within the investigated room are assumed as low-speed, incompressible flow. These assumptions were expressed in equation 1 and 2. The ANSYS Fluent software was employed as a prediction tool to simulate the air movement and temperature distribution for this study. The enhanced wall treatment was employed to solve the flow near the wall especially near the human body and near the heat source. This will give an accurate results of the air flow behavior in these interesting area. The assumption of Boussinesq was also used for the density calculations of this simulation. The SIMPLE algorithm was selected for the velocity and pressure coupling. Also, PRESTO was used in this simulation. The boundary conditions that used in this investigation are shown in figure 3.

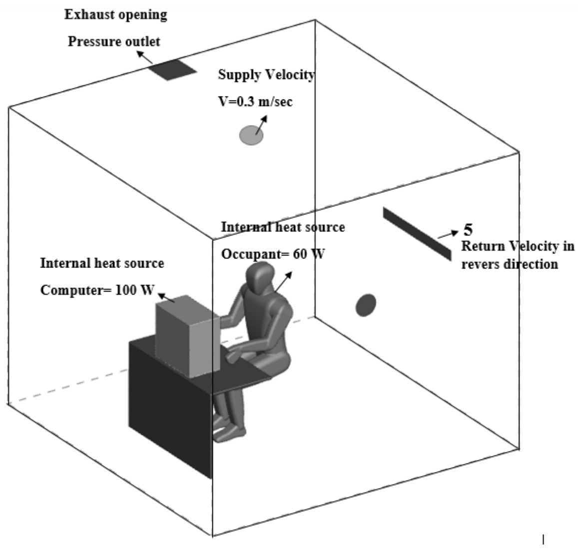


Fig. 3. The boundary condition in the simulated room

3. Validation

To demonstrate the ability and the accuracy of the CFD model that employed in this investigation, a validation work has been performed in this study. An experimental previous study performed by Cheng et al. [34] was used for this validation. In their study, five different positions of return opening were investigate in an examined room that employed a displacement ventilation as a main ventilation system to show the impacts of the return location on the human comfort and energy consumption. In this study, figure 4 present the comparison, at three poles (with different elevation from floor, between the experimental results from the previous work and simulation results. The comparison between the simulation results and experimental results show a good agreement between them. Depending on this comparison it is easy to say that this model can predict the air movement and temperature distribution correctly and accurately for the current simulation study. For more detailed information on the experimental work see reference [34]. Also, detailed information for the validated room are listed in Table 4.

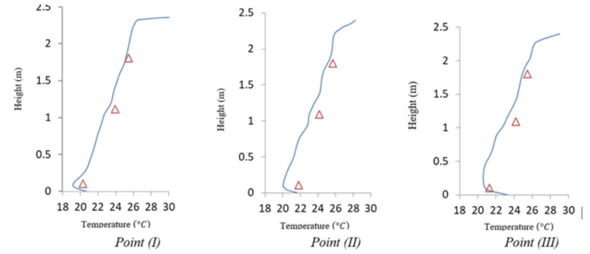


Fig. 4. The experimental and simulations comparison (triangle: experimental; continuous line: current simulation).

Table 4. Detailed information for the validated room.

Ceiling, floor, tables and bounded walls	Adiabatic wall
Supply air	Velocity inlet (84 l/s, 19 °C) and 1 × 0.6 m in dimension
Exhaust	Pressure –outlet and 0.45 × 0.2 m in dimension
Occupants	Uniform heat flux 80 W×2
Computer	Uniform heat flux 100 W ×2
Lighting	Uniform heat flux 24 W ×4
Data logger	25 W ×2
Room dimensions	4 (L) × 2.7 (W) × 2.4 (H)

4. Results and discussion

4.1 Indoor flow field evaluation

The indoor air flow behavior has an important influence on human comfort and IAQ. Therefore, it is very necessary to assess the indoor air velocity distribution for any ventilation system. In order to create a comfortable environment, the air velocity indoor should be taken carefully to satisfy the thermal comfort requirement [35]. This requirement were different from one application to other such as office room, hospital, gym and industrial room. Where for each application there are a recommended value of velocity. In this section, the influence of air velocity supplied by the proposed CPV system and its behavior were presented and discussed. The contour of the velocity air distribution at x=1.5 for case-1 and case-2 are shown in figures 5 and 6 respectively. From these figures, a relatively high velocity was found in case-2 especially in the area around the occupant, ranged between 0.35 to 0.45 m/sec, compared with case-1 (see figure 5 and 6). This was because that in case-1 the supply velocity was lower than in case 2. Also this differences means that the flow around the occupant influenced by the reached velocity to the occupant zone and this will may causes thermally discomfort area depending on the value of the supply velocity. However, an insulated zone around the occupant will created in this type of proposed ventilation system. This will protect the occupant from the unwanted contaminant air and provide a good inhaled fresh air because of this type of proposed ventilation system will directed the high quality of the supply air to the inhaled zone. In addition, the created canopy around the seated occupant was clearly presented in case-2 comparing with case-1 (see Figure 5 and 6). This was because the high supply velocity in case-2 will create a strong canopy around the occupant. Also, the air movement and the flow field behavior derived from the simulation results display an interesting flow fluctuation and high turbulence flow, especially in the working area around the occupant and also in the area close to the exhaust and return opening. It can be concluded that using the proposed PVC system with relatively high velocity such in case-2 will enhance the quality of the inhaled air and improve the energy consumption that required to provide a fresh air. However, the high air velocity in the occupant zone may increase the human thermal draught risk.

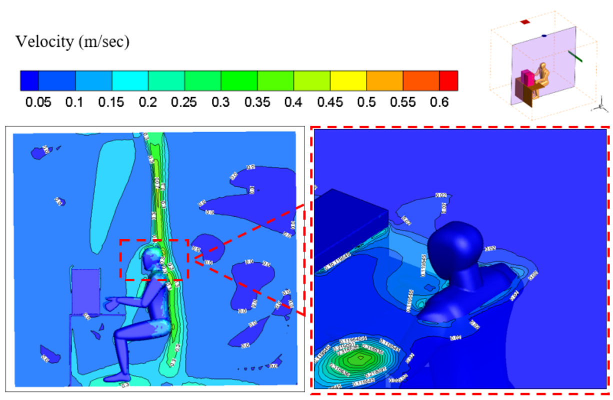


Fig. 5. Contour of air velocity distribution at plan x=1.5 m for case-1 (v=0.5 m/sec).

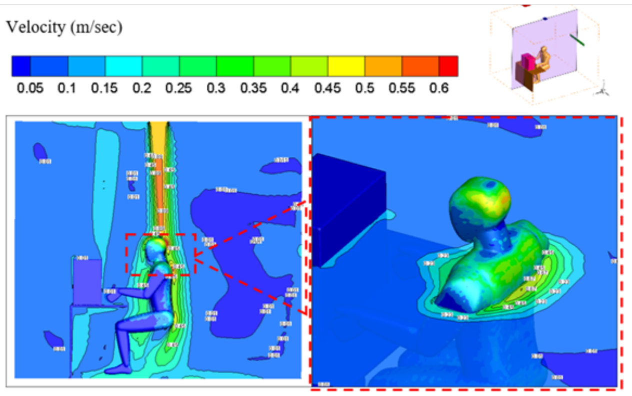


Fig. 6. Contour of air velocity distribution at plan x=1.5 m for case-2 (v=0.7 m/sec).

4.2 Temperature distribution indoor

The temperature distribution evaluations consider one of the main important assessment factors that have a great impact on providing the indoor thermal environment. The contour of the temperature distribution at x=1.5 for case-1 and 2 are shown in figures 7 and 8 respectively. From these figures, it is clear to see that the low air temperature values were found in case -2 compared with case-1 especially in the area near the occupant. This was because that in case -2 the supplied fresh air reached to the occupant zone faster than in case-1 due to the different in supply velocity for each case study. This will save more energy because the supply fresh air will create a thermally comfortable area around the occupant and this type of ventilation system will supplied a good quality air to the occupant zone directly before mixing with other warm air. This will reduce the load on the cooling coil and lead to save more energy. Also, for the both cases, case 1 and 2, the temperature in the rest of the room domain was relatively high comparing with the occupant area. This was because that this system targeted the occupant zone only with limited working hours. Therefore, it is not necessary to generate an ideal comfort condition for the unoccupied zone in the room domain. Also, the heat flux generated by the occupant and the computer, internal heat sources, will create a thermal plumes moved towards the ceiling. In opposite direction, the supply air from the ceiling will displaced the generated heat flux away from the occupant zone. This will also improve the quality of the inhaled air. From these finding, it can be concluded that using the proposed PVC system will enhance the air temperature distribution in area around the occupant and enhanced the energy consumption that demand by the cooling coil to provide the fresh air.

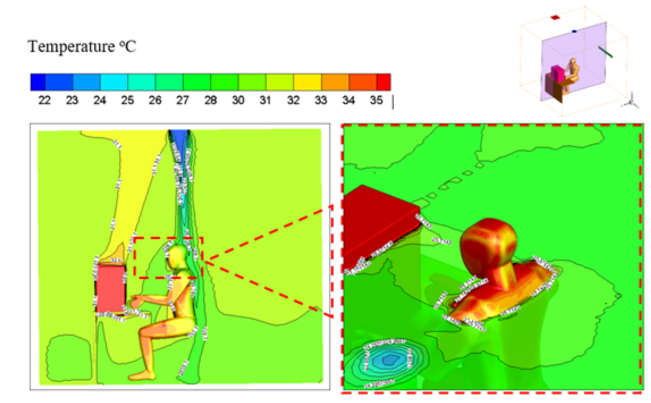


Fig. 7. Contour of air temperature distribution at plan x=1.5 m for case-1 (v=0.5 m/sec).

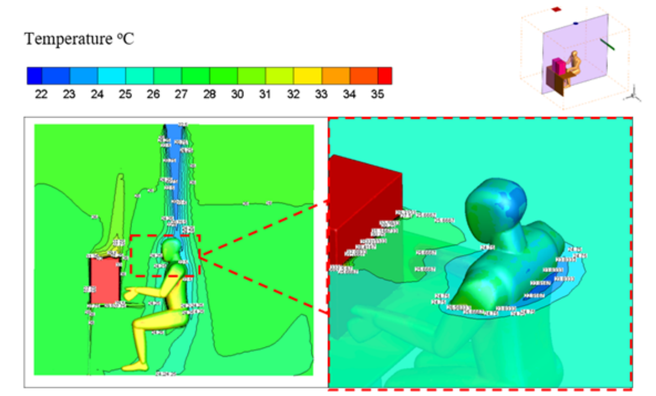


Fig. 8. Contour of air temperature distribution at plan x=1.5 m for case-2 (v=0.7 m/sec).

5. Conclusion

The aim of this work is to investigate the effect of employing the CPV system on energy saving and thermal comfort when using the CPV system as the main ventilation system. The CPV system was positioned at the center of the ceiling which is located above the occupant zone. This location will give the ability of the supplied fresh air to reach directly to the occupant zone. The results that obtained by the validated CFD model found that:

- Using CPV system will create an insulated zone bounded the occupant. A comfortable local environment and a high inhaled air quality was found in this local zone provided by the CPV system.
- Using the proposed PVC system with relatively high velocity, ranged between 0.05 to 0.15 m/sec., such in case-2 will improve the quality of the inhaled air and create a thermally comfortable zone.
- The proposed supply diffuser CPV system will provide the treated fresh directly to the occupant zone. Also, this was reduced the amount of the treated fresh air and enhance the energy consumed by the cooling coil. Moreover, using the proposed CPV system will create a homogenous thermal environment regarding the indoor distribution of the temperature, ranged between 22 to 30 ^{o} C, which may enhance the human thermal comfort and reduced the unwanted thermal effects.
- In case -2 the supplied air cover the occupant zone faster than in case-1. This was because the supply velocity for each case study was not same. This will improve the consumption of energy amount because the supply air will generate a thermally comfortable zone in area close to the occupant. Therefore, this type, CPV, of ventilation system will supplied a good quality of inhaled air to the occupant zone and save more energy.

Acknowledgements

The authors thank the Middle Technical University, Baghdad, Iraq to fund this project.

References

[1] R. F. Rupp, N. G. Vásquez, and R. Lamberts, "A review of human thermal comfort in the built environment," Energy and Buildings, vol. 105, pp. 178-205, 2015.

[2] L. Pérez-Lombard, J. Ortiz, and C. Pout, "A review on buildings energy consumption information," Energy and buildings, vol. 40, pp. 394-398, 2008.

[3] M. Veselý and W. Zeiler, "Personalized conditioning and its impact on thermal comfort and energy performance–A review," Renewable and Sustainable Energy Reviews, vol. 34, pp. 401-408, 2014.

[4] A. Muzet, J.-P. Libert, and V. Candas, "Ambient temperature and human sleep," Cellular and Molecular Life Sciences, vol. 40, pp. 425-429, 1984.

[5] S. H. Sheldon, R. Ferber, and M. H. Kryger, Principles and practice of pediatric sleep medicine: Elsevier Health Sciences, 2005.

[6] A. Q. Ahmed and S. Gao, "Numerical investigation of height impact of local exhaust combined with an office work station on energy saving and indoor environment," Building and Environment, vol. 122, pp. 194-205, 2017.

[7] A. Q. Ahmed, S. Gao, and A. K. Kareem, "Evaluation of thermal comfort, indoor air quality and energy saving of a Local Exhaust Ventilation system in an Office room (LEVO)."

[8] A. Q. Ahmed, S. Gao, and A. K. Kareem, "A numerical study on the effects of exhaust locations on energy consumption and thermal environment in an office room served by displacement ventilation," Energy Conversion and Management, vol. 117, pp. 74-85, 2016.

[9] A. Q. Ahmed, S. Gao, and A. K. Kareem, "Energy saving and indoor thermal comfort evaluation using a novel local exhaust ventilation system for office rooms," Applied Thermal Engineering, vol. 110, pp. 821-834, 2017.

[10] T. Hoyt, K. H. Lee, H. Zhang, E. Arens, and T. Webster, "Energy savings from extended air temperature setpoints and reductions in room air mixing," 2005.

[11] M. Chludzińska and A. Bogdan, "The effect of temperature and direction of airflow from the personalised ventilation on occupants' thermal sensations in office areas," Building and Environment, vol. 85, pp. 277-286, 2015.

[12] S. Hawendi, S. Gao, and A. Q. Ahmed, "Effect of heat loads and furniture on the thermal comfort of an isolated family house under a naturally ventilated environment," International Journal of Ventilation, pp. 1-26, 2019.

[13] N. Mahyuddin, H. B. Awbi, and E. A. Essah, "Computational fluid dynamics modelling of the air movement in an environmental test chamber with a respiring manikin," Journal of Building Performance Simulation, vol. 8, pp. 359-374, 2015.

[14] P. O. Fange, "Human requirements in future air-conditioned environments," in Advances in Building Technology, ed: Elsevier, 2002, pp. 29-38.

[15] F. F. Al-ajmi, "Thermal comfort in air-conditioned mosques in the dry desert climate," Building and Environment, vol. 45, pp. 2407-2413, 2010.

[16] M. Frontczak and P. Wargocki, "Literature survey on how different factors influence human comfort in indoor environments," Building and environment, vol. 46, pp. 922-937, 2011.

[17] M. J. Holmes and J. N. Hacker, "Climate change, thermal comfort and energy: Meeting the design challenges of the 21st century," Energy and Buildings, vol. 39, pp. 802-814, 2007.

[18] A. K. Melikov, "Personalized ventilation," Indoor air, vol. 14, pp. 157-167, 2004.

[19] B. Halvoňová and A. K. Melikov, "Performance of "ductless" personalized ventilation in conjunction with displacement ventilation: Impact of intake height," Building and Environment, vol. 45, pp. 996-1005, 2010.

[20] M. Taheri, M. Schuss, A. Fail, and A. Mahdavi, "A performance assessment of an office space with displacement, personal, and natural ventilation systems," in Building Simulation, 2016, pp. 89-100.

[21] S. Sekhar, N. Gong, K. W. Tham, K. Cheong, A. K. Melikov, D. Wyon, et al., "Findings of personalized ventilation studies in a hot and humid climate," HVAC&R Research, vol. 11, pp. 603-620, 2005.

[22] A. Makhoul, K. Ghali, and N. Ghaddar, "Low-mixing coaxial nozzle for effective personalized ventilation," Indoor and Built Environment, vol. 24, pp. 225-243, 2015.

[23] A. Makhoul, K. Ghali, and N. Ghaddar, "Desk fans for the control of the convection flow around occupants using ceiling mounted personalized ventilation," Building and Environment, vol. 59, pp. 336-348, 2013.

[24] I. Cruceanu, C. Maalouf, I. Colda, and M. Lachi, "Parametric study and energy analysis of a personalised ventilation system," Int J Math Models Meth Appl Sci, vol. 7, pp. 141-148, 2013.

[25] J. Kaczmarczyk, A. Melikov, Z. Bolashikov, L. Nikolaev, and P. O. Fanger, "Human response to five designs of personalized ventilation," HVAC&R Research, vol. 12, pp. 367-384, 2006.

[26] B. Yang, C. Sekhar, and A. K. Melikov, "Ceiling mounted personalized ventilation system in hot and humid climate—An energy analysis," Energy and Buildings, vol. 42, pp. 2304-2308, 2010.

[27] J. Pantelic, G. N. Sze-To, K. W. Tham, C. Y. Chao, and Y. C. M. Khoo, "Personalized ventilation as a control measure for airborne transmissible disease spread," Journal of the Royal Society Interface, vol. 6, pp. S715-S726, 2009.

[28] Z. D. Bolashikov and A. K. Melikov, "Methods for air cleaning and protection of building occupants from airborne pathogens," Building and Environment, vol. 44, pp. 1378-1385, 2009.

[29] K. Horikiri, Y. Yao, and J. Yao, "Numerical study of unsteady airflow phenomena in a ventilated room," in ICHMT digital library online, 2012.

[30] K. Horikiri, Y. Yao, and J. Yao, "Modelling conjugate flow and heat transfer in a ventilated room for indoor thermal comfort assessment," Building and Environment, vol. 77, pp. 135-147, 2014.

[31] J. Srebric and Q. Chen, "Simplified numerical models for complex air supply diffusers," Hvac&R Research, vol. 8, pp. 277-294, 2002.

[32] X. Yuan, Q. Chen, L. R. Glicksman, Y. Hu, and X. Yang, "Measurements and computations of room airflow with displacement ventilation," Ashrae Transactions, vol. 105, p. 340, 1999.

[33] V. Yakhot, S. Orszag, S. Thangam, T. Gatski, and C. Speziale, "Development of turbulence models for shear flows by a double expansion technique," Physics of Fluids A: Fluid Dynamics, vol. 4, pp. 1510-1520, 1992.

[34] Cheng Y, Niu J, Liu X, Gao N. Experimental and numerical investigations on stratified air distribution systems with special configuration: Thermal comfort and energy saving. Energy and Buildings. 2013;64:154-61..

[35] R. Yao, B. Li, and J. Liu, "A theoretical adaptive model of thermal comfort–Adaptive Predicted Mean Vote (aPMV)," Building and environment, vol. 44, pp. 2089-2096, 2009.