

NUMERICAL SIMULATION OF THE CO₂ DIFFUSION EFFECT ON LOW TURBULENT MIXED CONVECTION IN A VENTILATED ROOM HEATED BY THE BOTTOM

by

Lounes KOUFI^a, Zohir YOUNSI^b, and Hassane NAJI^{b*}

^aLMDC, INSA/UPS Genie Civil, Toulouse, France

^bUniversite Artois, Universite Lille, Yncrea-HEI, IMT-Douai, Bethune, France

Original scientific paper

<https://doi.org/10.2298/TSCI200116291K>

*A double-diffusive mixed convection within low turbulent regime in a ventilated cavities filled with an air CO₂ mixture and heated from below has been numerically investigated. The lower wall was sustained at a uniform temperature and CO₂ concentration. The vertical and upper walls were kept at external temperature and CO₂ concentration. To analyze the behavior of flow, the ventilation effectiveness for temperature distribution and removal of CO₂ contaminant, four configurations were dealt. These differ from each other by the location of the mixture inlet and outlet gaps. Likewise, three CO₂ concentrations were considered (10³, 2·10³, and 3·10³ ppm) to investigate the influence of the CO₂ diffusion on the ventilation effectiveness. The numerical simulations were performed by considering closed Reynolds averaged-Navier-Stokes equations using the Reynolds-normalization group *k-ε* model. The governing equations' set was then solved using the finite volume method, in which the pressure-velocity coupling was handled using the SIMPLEC algorithm. Validation of the numerical model was achieved by comparing our results with available experimental data. The obtained results indicate that the CO₂ diffusion effect on the air movement and the ventilation effectiveness for temperature distribution can be neglected in the present study. However, the CO₂ diffusion remains a key parameter in terms of indoor air quality index. Also, it was found that one of the studied configurations provides a better ventilation effectiveness to remove heat and CO₂ contaminant, and insures a homogeneous temperature and CO₂ concentration in the occupied zone. The three other configurations maintain an acceptable level of heat and can be used in temperate climate to ensure good indoor air quality.*

Key words: *heat transfer, turbulent mixed convection, contaminant diffusion, ventilated room, numerical simulation.*

Introduction

Over the last few decades, indoor air pollution has become a major concern and presents potential risks to public health. To deal with this situation, air renewal is one of the promising solutions. To achieve an efficient air renewal, it is essential to improve our knowledge of how ventilation distributes fresh air and how pollutants are transported within indoor spaces. Nowadays, indoor air quality (IAQ) has become of greater concern due to the long time periods people stay inside confined spaces. It has been mentioned that we spend more than 90% of our time in closed spaces such as homes, schools, offices, transports or shopping centers [1]. Fur-

* Corresponding author, e-mail: hassane.naji@univ-artois.fr

thermore, contaminant levels within indoor environment are typically much higher than those of outdoor environment [2]. More and more polluted products are being emitted by used building materials and furnishings, consumer and decoration products. Further, since the 1973 oil crisis, indoor spaces have been sealed and narrowed to reduce the energy consumption involved with heat losses through the envelopes [3]. Though such a development has reduced costs associated with energy consumption, it has also generated other issues such as confinement and poor air quality [3].

To avoid these constraints related to the air tightness, the ventilation is among the promising solutions. Its main objective is to provide a healthy environment by supplying unpolluted fresh air and clearing out contaminants. Moreover, the indoor air-flow is an important element for thermal comfort, which depends on several parameters such as air temperature, radiant temperature, humidity, air velocity, *etc.* Therefore, it is required to have an idea of how the fresh air is distributed indoors. Many investigations according to the characteristics of indoor air-flows so as to control temperature distribution and pollutant removal have been documented [4-9], to name a few.

In indoor environments, it is well known that two physical phenomena occur, namely the natural-convection generated by the buoyancy forces induced by temperature and pressure gradients, and the forced convection associated to the mechanical ventilation. The competition between buoyancy forces and external forces can generate mixed convection. Moreover, in the presence of concentration gradients, indoor air-flow becomes complex and is governed by double-diffusive mixed convection effects.

As is well known, double-diffusive mixed convection air-flows are encountered in many applications (*i.e.* engineering and geophysical). This includes room ventilation, nuclear reactors cooling, solar ponds, lakes and reservoirs, solar collectors, cooling of electronic devices, crystal growth [10-13]. Literature review highlights that various researchers have investigated the mixed convective flows in ventilated cavities, using analytical, experimental, and numerical approaches.

In what follows, we briefly describe some work related to the turbulent regime in ventilated cavities with contaminant gradients. Serrano-Arellano *et al.* [5] numerically conducted a study of heat and mass transfers induced by conduction, convection and radiation inside different ventilated cavities filled with an air-CO₂ mixture, which differ by the location of the mixture outlet opening. They reported that, in general, to obtain the advised values of temperature and CO₂ concentrations, it was sound to consider mixed convection at Reynolds number from 10⁴. Liu *et al.* [6] numerically performed a study of heat and mass transfer by turbulent mixed convection in a 3-D-ventilated cavity having an air conditioner window equipped with three slots for blowing, extracting and re-circulation. They stated that contaminant reduction rates can be achieved either by increasing fresh air ratio or Reynolds number, increasing the supply air-flow, or reducing the heating source. Cao and Meyers [7] have developed a fast approach based on low dimensional models for ventilation monitoring and control. They have shown that the flow equations decouple from the concentration equation when the mass air-flow/pollutant mass-flow ratio is sufficiently large. In another study [8], they investigated the turbulent inlet quantities effect on indoor air-flow parameters and pollutant dispersion using two turbulence models (low Reynolds number $k-\epsilon$ and SST $k-\omega$ models). They found that the turbulent length scales variation at the inlet induce small changes in the indoor flow field. In addition, regarding the pollutants dispersion, they indicated that the inlet turbulent quantities fluctuation leads to differences in pollutant concentration up to 20%. Deng *et al.* [9] numerically analyzed the air change rates effect on indoor CO₂ diffusion and removal inside a 3-D-ventilated room. The

authors stressed that the results obtained would be of great interest for practical applications to design and control ventilation systems from a health and energy efficiency perspective. Kračičik *et al.* [14] experimentally conducted a study on air distribution, ventilation effectiveness and thermal environment inside a room heated and ventilated by hot air supplied by a mixing ventilation system. Comparing with the case of underfloor heating, these authors found that the cooling supply air mixed well and created very uniform conditions with proper ventilation efficiency when the underfloor heating system was used. Rodriguez-Munoz *et al.* [15] analyzed the effect of heat and CO₂ emitted by a human being exhibited to mixed turbulent convection with radiation in a naturally ventilated room. They found that the natural ventilation reduces the average room temperature by about 4-5.5°C, while radiation rises it between 0.2 °C and 0.4 °C. Serrano-Arellano *et al.* [16] performed a study to define an optimal configuration based on the location of air outlet port allowing better CO₂ contaminant removal from 2-D-ventilated cavities with the air inlet port located on the lower right vertical wall. They found that the case where the air outlet port is at the top side removes a greater amount of contaminant for a $Re = 5 \cdot 10^3$, which had an effect on energy savings. In transient regime, Serrano-Arellano *et al.* [17] numerically studied double-diffusive convection with conduction and radiation conditions inside a ventilated cavity filled with an air-CO₂ mixture where an air outlet gap was set close to the heat and contaminant sources. They reported that the advised thermal and air quality parameters were met within 15 seconds for $Re = 1 \cdot 10^4$.

Based on the literature review up-reported, it appeared that research on the ventilation systems' ability to ensure better temperature distribution and better CO₂ diffusion in environments ventilated interiors operating with low Reynolds numbers remain scarce. Thereby, it seems useful to conduct a numerical study to predict the air-flow behavior inside ventilated rooms subjected to temperature and concentration gradients as a first approximation. The objectives here are twofold:

- to numerically perform a study of the low turbulent double-diffusive mixed convection from a ventilated cavity heated from below and
- to assess the CO₂ diffusion effect on the air-flow behaviour, the temperature and concentration distributions, and the IAQ.

Computational analysis

Physical domain

Figure 1 shows the investigated configuration ($1.04 \times 1.04 \text{ m}^2$, *i.e.* length and height) with the Cartesian co-ordinate system. We consider this cavity because of the interesting experimental data availability [18]. Here, the working fluid is an air-CO₂ mixture. The bottom wall is subjected to uniform temperature, T_H , and CO₂ concentration, C_H . External conditions (*i.e.* temperature, T_C , and CO₂ concentration, C_L) are applied to the other walls, with $T_H > T_C$ and $C_H > C_L$. The air-CO₂ mixture is renewed by providing fresh air from outside, T_C and C_L , through an opening mounted at the top of the left wall. The polluted air leaves the cavity through an opening at the bottom of the

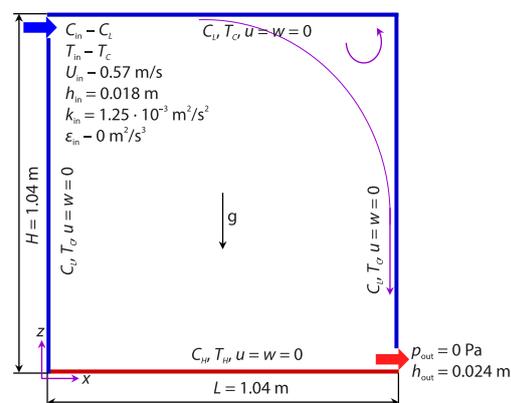


Figure 1. Ventilated cavity configuration with associated boundary conditions

right side. The air inlet velocity is fixed at 0.57 ms^{-1} , which leads to $Re = 706$. The considered thermal Rayleigh number is $2.62 \cdot 10^9$ (with $T_H = 308.5 \text{ K}$). The Lewis number corresponding to considered thermal and mass diffusivities, α and D , is equal to 1.47. The thermophysical properties of air-CO₂ mixture were achieved by using recommended relationships by Poling *et al.* [19].

Governing equations

To analyze the steady-state low turbulent double-diffusive flow in the whole cavity, the commercially available CFD software scStream was used. The following assumptions were made-up:

- the flow is turbulent, fully developed and viscous,
- the air-CO₂ mixture is Newtonian and incompressible under the Boussinesq approximation, and
- the dissipation, pressure forces' work, and radiation are assumed to be negligible. In addition, the CO₂ concentration level is low (it is 0.04% lower compared to air).

The RANS equations coupled to the RNG k - ε turbulence model having been deemed herein are those described in [20].

Boundary conditions

The appropriate boundary conditions for the present problem are: $u = w = 0$ on the all solid surfaces, $u = u_{in}$ and $w = 0 \text{ m/s}$ at the air inlet, $\partial(u, w)/\partial n = 0$ at the air outlet, n being the unit normal. As for thermal boundary conditions, the temperature of the: air inlet is $T_{in} = 288 \text{ K}$, floor is $T_H = 308.5 \text{ K}$, other walls is $T_C = 288 \text{ K}$. For the CO₂ contaminant, at inlet port is $C_{in} = 350 \text{ ppm}$, at the floor is C_H (varied between 10^3 and $3 \cdot 10^3 \text{ ppm}$), at the other walls is $C_L = 350 \text{ ppm}$. The boundary conditions for T and C at the outlet are Neumann conditions, *i.e.*, $\partial T/\partial n = \partial C/\partial n = 0$. Regarding the turbulent quantities, their values are $k_{in} = 1.25 \cdot 10^{-3} \text{ m}^2/\text{s}^2$ and $\varepsilon_{in} = 0 \text{ m}^2/\text{s}^3$ at the inlet port [18], while at the air outlet, $\partial k/\partial n = \partial \varepsilon/\partial n = 0$. At this port (outlet), an atmospheric pressure value has been set.

Numerical procedure

Discretization

The aforementioned governing equations were solved numerically employing a finite-volume method. To avoid possible artificial diffusion, which is caused by truncation errors due to the discretization, a QUICK scheme and a second-order central differencing scheme were involved in approximating the advection and diffusion terms, respectively. Note that the pressure-velocity coupling was performed using the SIMPLEC algorithm, and that the numerical resolution was achieved using multiple-iteration constrained conjugate gradient. When the residual values of the different equations are sufficiently low for every control volume, overall convergence is achieved (10^{-8} for each variable). Table 1 shows the results' convergence of heat balance at the last iteration of the computation. We find that the receiving and release heat are equivalent meaning, thereby, that convergence is reached.

Table 1. Heat receiving and release balances for the computation's last iteration with $C_H = 3 \cdot 10^3 \text{ ppm}$

Heat receiving balance [W]	Heat release balance [W]	Total [W]
$2.48 \cdot 10^{-2}$	$2.48 \cdot 10^{-2}$	0.00

To ensure that the results are grid-independent and well-resolved, the simulations were repeated with different fine levels of meshing. It was determined that there were no noticeable differences in the solutions when the number of cells is around 196×196 cells or greater. Table 2 illustrates the relative error of mean temperature and concentration, T_m and C_m , maximum and minimum velocity components (at $X = 0.5$ or $Z = 0.5$), and their locations, X or Z , of the 196×196 grid. A maximum deviation of 1.0% for all variables between meshes 196×196 and 216×216 grid was spotted.

Table 2. Relative errors between grids and 196×196 grid with $C_H = 3000$ ppm

Grid	176×176	196×196	216×216
C_m [ppm]	836.06 (0.66%)	830.59	826.76 (0.46%)
T_m [K]	292.47 (0.00%)	292.47	292.47 (0.00%)
$u_{max} \cdot 10^{-1}$ at $X = 0.5$ [ms^{-1}]	3.42 (1.72%)	3.48	3.51 (0.86%)
Z	0.99 (0.70%)	0.99	0.99 (0.00%)
$u_{min} \cdot 10^{-1}$ at $X = 0.5$ [ms^{-1}]	-3.57 (0.56%)	-3.59	-3.59 (0.00%)
$Z \cdot 10^{-2}$	1.83 (0.00%)	1.83	1.84 (0.55%)
$w_{max} \cdot 10^{-1}$ at $Z = 0.5$ [ms^{-1}]	3.09 (1.59%)	3.14	3.16 (0.64%)
$X \cdot 10^{-2}$	3.22 (6.27%)	3.03	3.00 (0.99%)
$w_{min} \cdot 10^{-1}$ at $Z = 0.5$ [ms^{-1}]	-3.88 (0.51%)	-3.90	-3.91 (0.26%)
X	0.99 (0.00%)	0.99	0.99 (0.00%)

Numerical validation

To check and validate our approach, the ventilated cavity under thermal buoyancy force and low turbulent regime reported by [18] was solved. To avoid overloading the text, note that the verification and validation processes have already been presented in previous work [21-24], with satisfactory results. Likewise, the check process was completed via the problem addressed by Ampofo and Karayiannis [25]. It consists on a 2-D-ended square cavity filled with air and differentially heated between the vertical walls in turbulent regime. To deal with such a benchmark case, the following dimensionless parameters were used: $Pr = 0.71$ and $Ra_T = 1.58 \cdot 10^9$. The results are shown in fig. 2 for dimensionless temperature, w -velocity plotted at $Z = 0.5$, and local Nusselt number at the cold and hot walls. It can be seen that the results match those obtained experimentally [25]. Under these comparisons, we can conclude that the implemented approach shows satisfactory results.

Results and discussions

In the following, we present the results of 2-D-ventilated square cavity heated from below, and filled with an air-CO₂ mixture ($Pr = 0.71$). The heights of the air inlet and outlet openings are $h_{in} = 0.018$ m and $h_{out} = 0.024$ m. The air-CO₂ mixture penetrates at 288 K and 350 ppm with a velocity of 0.57 m/s. The vertical walls and the upper one are maintained at a uniform temperature and CO₂ concentration of 288 K and 350 ppm, respectively. The lower wall is attached a constant temperature ($T_H = 308.5$ K) and CO₂ concentration ($C_H > 350$ ppm). The obtained Reynolds and thermal Rayleigh numbers are equal to $2.62 \cdot 10^9$ and 706, respectively. It is useful to recall that, in the case of displacement ventilation, forced convection is generally low. Buoyancy forces due to the difference of temperature and concentration affect

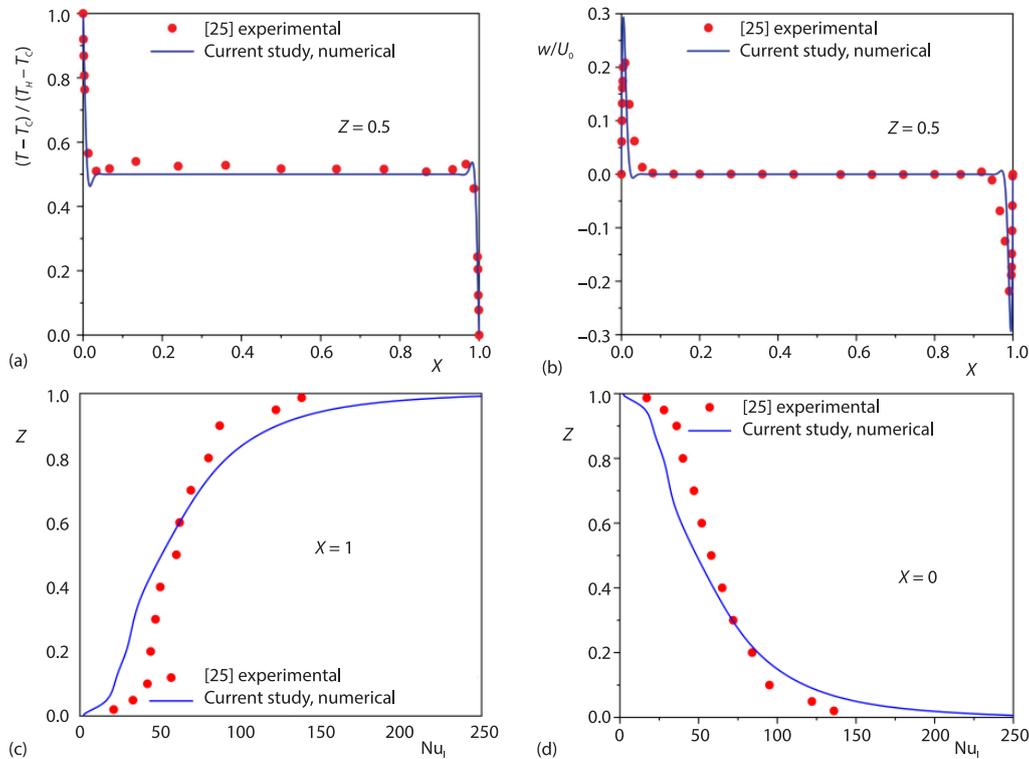


Figure 2. Comparison with [25]; (a) dimensionless temperature, (b) dimensionless w -velocity, (c) Nu_1 at the cold wall, and (d) Nu_1 at the hot wall with $Ra_7 = 1.58 \cdot 10^9$

the flow. The presence of these forces creates mixed convection which can occur with different magnitudes. To quantify this, the Richardson number that represents the ratio between the natural and forced convections ($Ri = (Ra_T/Pr)/Re^2$, where Ra_T and Re being defined with the same scale L) was defined. When $Ri = 1$, the two forces are equivalent and the developed flow is mixed convective. Depending on the range wherein the problem is defined $0 < Ri < 1$ or $1 < Ri < \infty$, the natural and forced convection forces are present with a different interaction. For $Ri \gg 1$, the flow is only driven by the free convection, while for $Ri \ll 1$, forced convection prevails. From the obtained Richardson number ($Ri = 2.23$), the flow is mixed convective with a slight dominance of free convection.

Flow patterns

To identify the flow behavior, and the distribution of temperature and CO₂ concentration inside the cavity, it is important to know how the fresh air is distributed in the cavity. For this, we illustrate isotherms, CO₂ isolines, streamlines and age of air in fig. 3 for different values of C_H (1000, 2000, and 3000 ppm). It should be noted that the corresponding buoyancy ratios, N , are $-3.12 \cdot 10^{-3}$, $-7.93 \cdot 10^{-3}$, and $-1.25 \cdot 10^{-2}$, respectively. It is worth noting that the flow is opposing when N is negative. Simply stated, the solutal buoyancy force opposes that of thermal buoyancy force.

Regarding isotherms, it is observed that the cold air becomes warm as it comes near the bottom. Then, it rises towards the cavity upper part. During this movement, the heat is

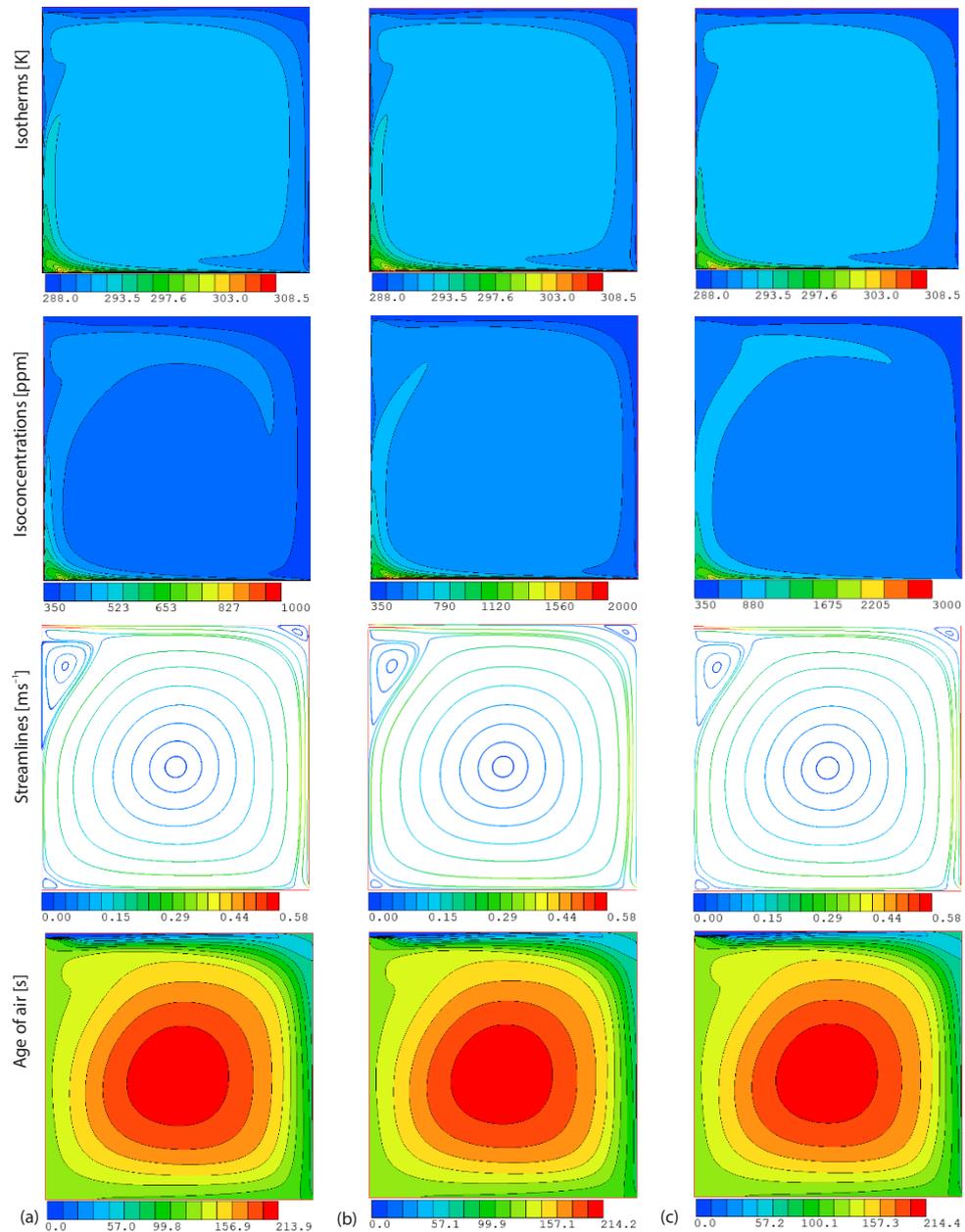


Figure 3. Isotherms [K], iso-concentrations [ppm], streamlines [ms⁻¹] and age of air [s]; (a) C_H = 1000 ppm, (b) C_H = 2000 ppm, and (c) C_H = 3000 ppm

trapped in the center of the cavity with some thermal stratification in the heart of the cavity while explaining the resting state of the fluid in this area. It should be noted that the isotherms are insensitive to the increasing of the CO₂ source, C_H, due to the low values of buoyancy ratios. This means the solutal buoyancy forces are negligible past the thermal buoyancy and external

forces are the motor of the motion. As for the CO₂ isolines, they follow the isotherms but the quantity of CO₂ increases as C_H growing. For streamlines, we observe a large re-circulation vortex in the cavity, which is driven by the jet and buoyancy forces. In addition, two smaller structures are distinguished in the upper right corner and the lower left one. The flow pattern for three C_H values appears to be almost identical for the same reason mentioned previously. Also, we note the emergence of a third cell close to the inlet port and below the jet. This is due to the Coanda effect and interaction between natural and forced convections. To get an idea about the time what fresh air takes to sweep the cavity, the local age of air [26] is presented. We found that fresh air takes less time to get closer to walls due to the closure of the flow at these locations. In the middle of the cavity, air passes over time due to prevailing low velocities at the center of cavity. Regarding the results obtained, there was no change in the air-flow and thermal behavior when C_H increases. This is due to the low contribution of the solutal buoyancy forces (the ratio N being close to zero).

To determine the value of C_H for which the flow physics is affected by solutal convection, we have performed a numerical experiment by increasing C_H to $2 \cdot 10^4$ ppm. Figure 4 illustrates the temperature, u - and w -velocity components, and intensity of turbulent kinetic energy profiles along the vertical and horizontal mid-planes for C_H equal to 10^3 , $3 \cdot 10^3$, and $20 \cdot 10^3$ ppm. We observe that the velocity components and temperature do not change with C_H . Although, the intensity of the turbulent kinetic energy is slightly sensitive when $C_H = 20 \cdot 10^3$ ppm. This confirms what has been said previously. As conclusion, the effect of solutal convection is negligible in the building scale due to the low CO₂ concentrations, but remains dangerous to human being health beyond a certain threshold concentration (10^3 ppm, in France). It may be advisable to consider the effect of CO₂ density in industrial processes where concentrations are much higher (*e.g.* combustion process).

The analysis of fig. 5 shows that the increasing of C_H causes an increase of the CO₂ concentration in the large central part of the cavity. It ranges from about 430 ppm for $C_H = 10^3$ ppm to 850 ppm when $C_H = 3 \cdot 10^3$ ppm. The CO₂ contaminant profiles plotted along the cavity bisector indicate that the CO₂ is homogeneous in the occupied zone and all concentration gradients are located near-walls. This demonstrates that there is no concentration pic that can harm occupants' health.

Table 3 shows the mean temperature, T_m , and CO₂ concentration, C_m , and their standard deviation, σ , within the cavity according to the C_H . We see clearly that T_m is insensitive to the increasing of C_H ($\sigma = 0$ K). In the other hand, CO₂ concentration is highly increased by C_H , σ is important (201.5 ppm). This is due to the increasing of the CO₂ source amount. The standard deviation σ is obtained via the relationship:

$$\sigma = \sqrt{\sum_{i=1}^N \frac{1}{N} (\phi_i - \bar{\phi})^2} \quad (1)$$

where ϕ_i is the discrete variable, $\bar{\phi}$ – the average value, and N – the number of observations.

Table 3. Mean internal temperature and CO₂ concentration and their standard deviation σ

C_H [ppm]	T_m [K]	C_m [ppm]
$1 \cdot 10^3$	292.5	427.5
$2 \cdot 10^3$	292.5	631.3
$3 \cdot 10^3$	292.5	830.6
σ	0.0	201.5

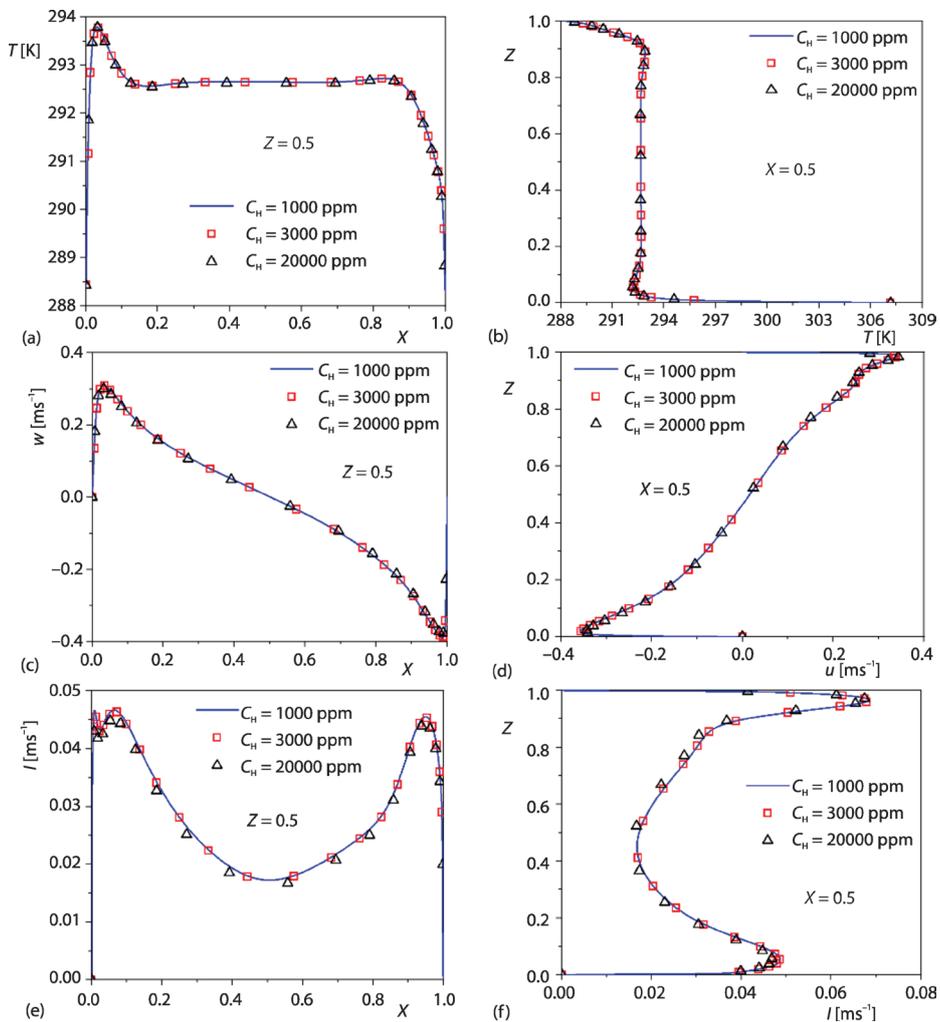


Figure 4. Contour plots along the horizontal and vertical cavity bisectors; (a, b) temperature, (c, d) w - and u -profiles, and (e, f) turbulent kinetic energy intensity ($I = k^{1/2}$), with different values of C_{H_2} ; (a, c, e) $Z = 0.5$ and (b,d,f) $X = 0.5$

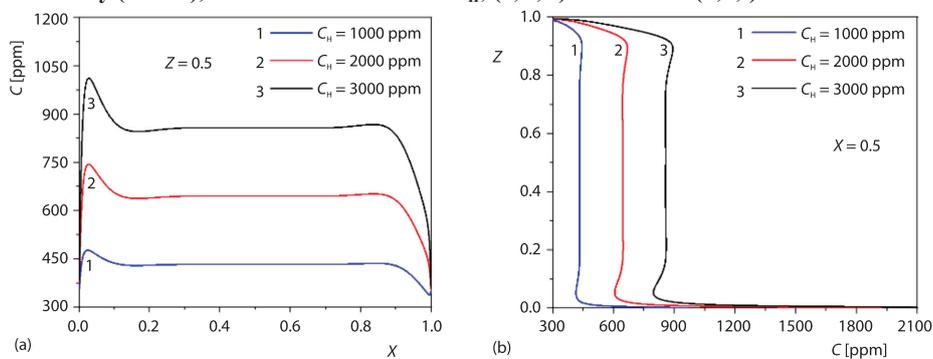


Figure 5. The CO₂ concentration plots along the cavity bisectors; (a) $Z = 0.5$ and (b) $X = 0.5$

Nusselt and Sherwood numbers on the hot wall

To quantify the convective heat and mass transfers, one often uses the local and average Nusselt and Sherwood numbers. Here, both are defined:

$$\text{Nu}_m = \frac{1}{L} \int_0^L (\text{Nu}_1)_{z=0} dx = \frac{-1}{L\Delta T} \int_0^L \left(\frac{\partial T}{\partial z} \right)_{z=0} dx \quad (2)$$

$$\text{Sh}_m = \frac{1}{L} \int_0^L (\text{Sh}_1)_{z=0} dx = \frac{-1}{L\Delta C} \int_0^L \left(\frac{\partial C}{\partial z} \right)_{z=0} dx \quad (3)$$

The local and averaged Nusselt and Sherwood numbers on the hot wall (bottom wall) for a wide range of C_H graphically depicted in fig. 6. We observe that, at the outside of the vertical boundary-layers, Nu_1 is minimum near the left wall (air-CO₂ mixture is hot) and increases to its maximum at the opposite wall (air-CO₂ mixture is cold), see fig. 6(a). As a function C_H , Nu_1 is identical because the variations of buoyancy ratio are very low. Regarding the local Sherwood number, fig. 6(b), Sh_1 has a similar evolution compared to Nu_1 with a slightly higher values and decreases softly when C_H increases. As for average Nusselt and Sherwood numbers, fig. 6(c), Nu_m remains stable when C_H increases, in contrary Sh_m which decreases. Based on these results, we conclude that solutal buoyancy forces do not contribute to the enhancement of heat and mass transfers.

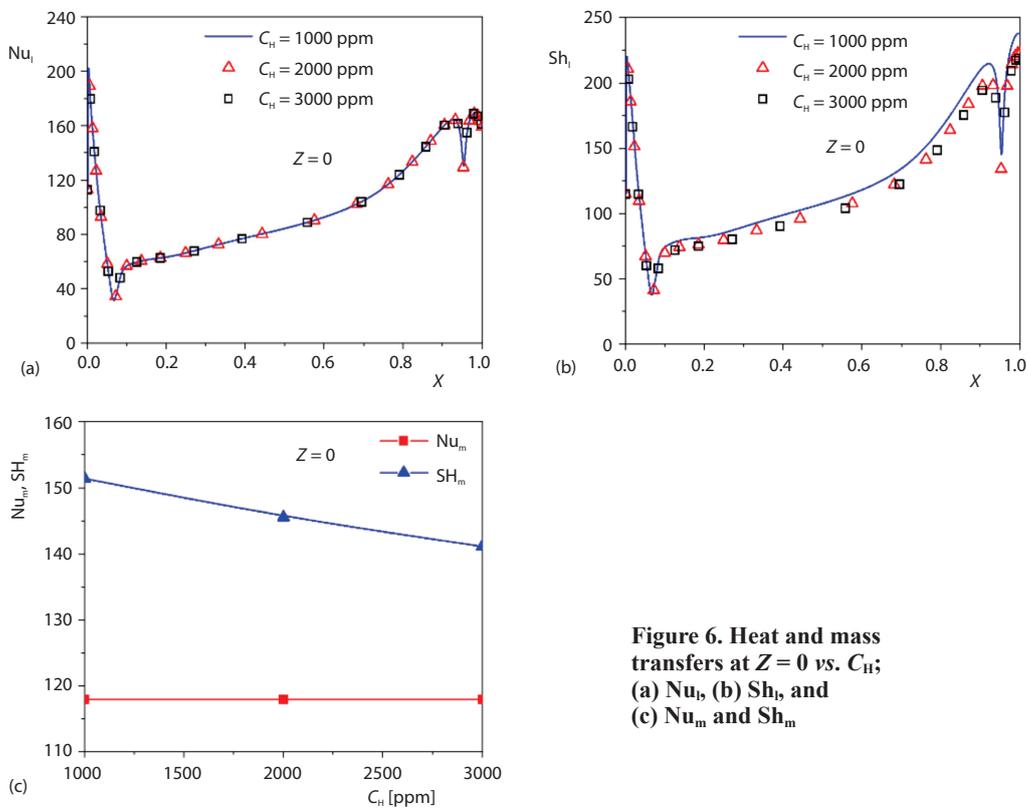


Figure 6. Heat and mass transfers at $Z = 0$ vs. C_H ; (a) Nu_1 , (b) Sh_1 , and (c) Nu_m and Sh_m

Characteristic parameters

In this section, the ventilation effectiveness for temperature distribution, ε_T , and removal of CO₂ pollutant, ε_C , and index of indoor air quality, I_{IAQ} , are analyzed for the mentioned parameters and boundary conditions. These parameters are relevant to get an information on the ventilation effectiveness to ensure the occupants comfort. They provide a quantitative index related to the way in which heat and pollutants are distributed all over inside the cavity, the higher the value is, the more homogeneous temperature and pollutants distributions are. Awbi expresses this parameter [27]:

$$\varepsilon_T = \frac{T_{out} - T_{in}}{T_m - T_{in}} \quad (4)$$

$$\varepsilon_C = \frac{C_{out} - C_{in}}{C_m - C_{in}} \quad (5)$$

$$I_{IAQ} = \frac{C_m - C_{out}}{C_{th} - C_{out}} \quad (6)$$

Four configurations were considered: Configuration A (inlet gap on the upper of the left wall and outlet on the lower of opposite wall, fig. 1, Configuration B (inlet gap on the lower of the left wall and outlet on the upper of the opposite wall), Configuration C (inlet opening on the lower part of the left side and outlet on the upper of the same side), and Configuration D (inlet on the upper of the left side and outlet on the lower of the same side). The obtained results are gathered in tab. 4 vs. C_H .

The results show that ε_T is not affected by the increase of C_H for the reasons up-mentioned indicating that solutal buoyancy forces do not induce changes in heat and mass transfers. Regarding the pollutant removal effectiveness, it is found that ε_C evolves with C_H . It was found that the Configuration D provides the highest value of ε_T and ε_C compared to the others. As can be seen, the values remain significantly larger than unity indicating that the ventilation sufficiently evacuate the heat and the CO₂ contaminant. In terms of indoor air quality, I_{IAQ} is less than unity for all configurations and C_H values considered meaning they all ensure good air quality. Nevertheless, the Configuration D provides the best performance. It is worthy to note that the sign (-) indicates that the CO₂ concentration at the outlet gap, C_{out} , is greater than the average CO₂ concentration of the cavity, C_m .

Based on the obtained results, it may be concluded that the Configurations A, B, and C uphold ε_T and ε_C close to unity and I_{IAQ} less than unity. Thereby, they can be used in temperate climate zones (e.g., Mediterranean climate) to ensure an acceptable IAQ and maintaining an appreciable temperature level. As for the Configuration D, it can be used to remove the internal heat while contributing to refresh the room in summer and insure a good IAQ.

Conclusions

The low turbulent double-diffusive mixed convection flow in a 2-D-ventilated room filled with an air-CO₂ mixture was computationally investigated. The emphasis has been on the CO₂ diffusion effect on the flow behavior, temperature distribution, and indoor air quality. Computations have been performed with thermal Rayleigh number of $2.62 \cdot 10^9$, Reynolds numbers of 706, Prandtl number of 0.71, Lewis number of 1.47, and various CO₂ source between 10^3 and $3 \cdot 10^3$ ppm. In RANS equations, the RNG $k-\varepsilon$ turbulence model was used. Based on the simulation results, our main findings can be drawn as follows.

Table 4. The ε_T , ε_C , and I_{IAQ} values vs. C_H

C_H [ppm]	ε_T	ε_C	I_{IAQ}	Configuration
$1 \cdot 10^3$	0.71	0.63	0.05	
$2 \cdot 10^3$	0.71	0.72	0.18	
$3 \cdot 10^3$	0.71	0.73	0.45	
$1 \cdot 10^3$	0.81	1.10	-0.01	
$2 \cdot 10^3$	0.81	1.14	-0.05	
$3 \cdot 10^3$	0.81	1.24	-0.23	
$1 \cdot 10^3$	0.73	0.94	0.00	
$2 \cdot 10^3$	0.73	1.09	-0.03	
$3 \cdot 10^3$	0.73	1.14	-0.13	
$1 \cdot 10^3$	4.68	5.79	-0.16	
$2 \cdot 10^3$	4.68	8.86	-0.96	
$3 \cdot 10^3$	4.69	11.79	-6.56	

- The analysis of the effect of CO₂ diffusion showed that solutal convection can be neglected in the field of building because the CO₂ concentration level is low but still dangerous. It is recommended to consider its effect on industrial applications where concentrations are heavy.
- Regarding heat and mass transfers, the increasing of CO₂ amount does not improve or de-mean both Nusselt and Sherwood numbers due to the low buoyancy ratios, which are closely zero.
- In terms of thermal comfort, the Configuration D exhibits a much higher ε_T (>unit) than that of other configurations. Thereby, it seems to be intended to evacuate sufficient internal heat, and may be suitable for use in summer. As for Configurations A, B, and C, they present a ε_T close to unity and can, therefore, be used in winter in temperate climates for the CO₂ release.
- Finally, from the results of the ventilation efficiency for the removal of pollutant, (ε_C , and the indoor air quality index, I_{IAQ}), it was found that the mixing ventilation process uphold an acceptable and homogeneous CO₂ concentration distributions, and that all configurations ensure an appreciable indoor air quality based on the CO₂ pollutant.

Nomenclature

C – chemical species' concentration, [kgm⁻³ or ppm]
 ΔC – characteristic concentration difference, (= $C_H - C_L$), [kgm⁻³]
 C_{Th} – threshold concentration, [kgm⁻³ or ppm]
 D – diffusion coefficient, [m²s⁻¹]
 g – gravity acceleration, [ms⁻²]
 H – cavity's height, [m]
 h – height of the inlet or outlet air gap, [m]
 I – turbulent kinetic energy intensity, [ms⁻¹]
 I_{AQ} – indoor air quality index, [-]
 k – turbulent kinetic energy, [m²s⁻²]
 L – cavity's width, [m]
 Le – Lewis number, (= α_0/D)
 N – buoyancy ratio, (= $\beta_s \Delta C / \beta_T \Delta T$)
 Nu – Nusselt number, [-]
 Pr – Prandtl number, (= ν_0/α_0)
 p – fluid pressure, [Pa]
 Ra_T – Rayleigh number, [= $g\beta_T \Delta TH^3 / (\nu_0 \alpha_0)$]
 Re – Reynolds number, (= $\rho_0 U_{in} h_m / \mu_0$)
 Ri – Richardson number, [= $(Ra_T/Pr)/Re^2$]
 Sh – Sherwood number, [-]
 T – fluid temperature, [K]
 ΔT – characteristic temperature difference, (= $T_H - T_C$), [K]
 U – velocity modulus, [ms⁻¹]
 U_0 – buoyancy velocity, [= $(g\beta_T \Delta TH)^{1/2}$], [ms⁻¹]
 u, w – horizontal and vertical velocities, [ms⁻¹]
 X, Z – dimensionless co-ordinates, [= $(x, z)/H$]
 x, z – horizontal and vertical co-ordinates, [m]

Subscripts

C – cold
 H – high or hot
 in – inlet
 L – low
 l – local
 m – mean or average
 max – maximum
 min – minimum
 S – solutal
 out – outlet
 0 – reference
 T – thermal

Greek symbols

α – thermal diffusivity, [m²s⁻¹]
 β_s – solutal expansion coefficient, [m³kg⁻¹]
 β_T – thermal expansion coefficient, [K⁻¹]
 ε – turbulence energy dissipation, [m²s⁻³]
 ε_C – ventilation effectiveness for pollutants removal, [-]
 ε_T – ventilation effectiveness for temperature distribution, [-]
 μ – dynamic viscosity, [kgm⁻¹s⁻¹]
 ν – kinematic viscosity, [m²s⁻¹]
 ρ – density, [kgm⁻³]
 σ – standard deviation

Acknowledgment

The authors would like to thank the anonymous reviewers for their insightful comments and suggestions which helped improve the manuscript initial version.

References

- [1] Deng, Q.-H., et al., Fluid, Heat and Contaminant Transport Structures of Laminar Double-Diffusive Mixed Convection in a 2-D Ventilated Enclosure, *International Journal of Heat and Mass Transfer*, 47 (2004), 24, pp. 5257-5269
- [2] Marchand, C., et al., Aldehyde Measurements in Indoor Environments in Strasbourg (France), *Atmospheric Environment*, 40 (2006), 7, pp. 1336-1345
- [3] Kabir, E., et al., Indoor Air Quality Assessment in Child Care and Medical Facilities in Korea, *Environmental Monitoring and Assessment*, 184 (2012), 10, pp. 6395-6409
- [4] Xaman, J., et al., Effect of a Contaminant Source (CO₂) on the Air Quality in a Ventilated Room, *Energy*, 36 (2011), 5, pp. 3302-3318
- [5] Serrano-Arellano, J., et al., Optimum Ventilation Based on the Ventilation Effectiveness for Temperature and CO₂ Distribution in Ventilated Cavities, *International Journal of Heat and Mass Transfer*, 62 (2013), 1, pp. 9-21
- [6] Liu, D., et al., Turbulent Transport of Airborne Pollutants in a Residential Room with a Novel Air Conditioning Unit, *International Journal of Refrigeration*, 35 (2012), 5, pp. 1455-1472
- [7] Cao, S.-J., Meyers, J., On the Construction and Use of Linear Low Dimensional Ventilation Models, *Indoor Air*, 22 (2012), 5, pp. 427-441

- [8] Cao, S.-J., Meyers, J., Influence of Turbulent Boundary Conditions on RANS Simulations of Pollutant Dispersion in Mechanically Ventilated Enclosures with Transitional Slot Reynolds Number, *Building and Environment*, 59 (2013), Jan., pp. 397-407
- [9] Deng, H.-Y., *et al.*, Influence of Air Change Rates on Indoor CO₂ Stratification in Terms of Richardson Number and Vorticity, *Building and Environment*, 129 (2018), Feb., pp. 74-84
- [10] Saha, S., *et al.*, Double Diffusive Mixed Convection Heat Transfer Inside a Vented Square Cavity, *Chemical Engineering Research Bulletin*, 13 (2009), 1, pp. 17-24
- [11] Abidi A., *et al.*, Effect of Heat and Mass Transfer through Diffusive Walls on 3-D Double-Diffusive Natural-Convection, *Numerical Heat Transfer*, 53 (2008), Part A, pp. 1357-1376
- [12] Abidi A., *et al.*, Effect of Radiative Heat Transfer on 3-D Double Diffusive Natural-Convection, *Numerical Heat Transfer*, 60 (2011), Part A, pp. 785-809
- [13] Ahmed K. H., *et al.*, The 3-D Unsteady Natural-Convection and Entropy Generation in an Inclined Cubical Trapezoidal Cavity with an Isothermal Bottom Wall, *Alexandria Engineering Journal*, 55 (2016), 2, pp. 741-755
- [14] Krajčik M., *et al.*, Air Distribution and Ventilation Effectiveness in an Occupied Room Heated by Warm Air, *Energy and Buildings*, 55 (2012), Dec., pp. 94-101
- [15] Rodriguez-Munoz, N. A., *et al.*, Numerical Study of Heat Transfer by Convection and Thermal Radiation in a Ventilated Room with Human Heat Generation and CO₂ Production, *Latin American Applied Research*, 43 (2013), 4, pp. 353-361
- [16] Serrano-Arellano, J., *et al.*, Indoor Air Quality Analysis Based on the Ventilation Effectiveness for CO₂ Contaminant Removal in Ventilated Cavities, *Revista Mexicana Física*, 60 (2014), 4, pp. 309-317
- [17] Serrano-Arellano, J., *et al.*, Numerical Investigation of Transient Heat and Mass Transfer by Natural-Convection in a Ventilated Cavity: Outlet Air Gap Located Close to Heat Source, *International Journal of Heat and Mass Transfer*, 76 (2014), Sept., pp. 268-278
- [18] Blay, D., *et al.*, Confined Turbulent Mixed Convection in the Presence of a Horizontal Buoyant Wall Jet, *ASME Fundamentals of Mixed Convection HTD*, 213 (1992), pp. 65-72
- [19] Poling, B. E., *et al.*, *The Properties of Gases and Liquids*, 5th ed., McGraw-Hill, New York, USA, 2001
- [20] Koufi, L., *et al.*, Numerical Investigation of Turbulent Double-Diffusive Mixed Convection in a Slot Ventilated Enclosure with Supply Air-Flow Ports, *Proceedings*, 23rd French Congress of Mechanics, Lille, France, 2017
- [21] Koufi, L., *et al.*, Numerical Investigation of Turbulent Mixed Convection in an Open Cavity: Effect of Inlet and Outlet Openings, *International Journal of Thermal Sciences*, 116 (2017), June, pp. 103-117
- [22] Koufi, L., *et al.*, Double-Diffusive Natural-Convection in a Mixture-Filled Cavity with Walls' Opposite Temperatures and Concentrations, *Heat Transfer Engineering*, 40 (2018), 15, pp. 1-18
- [23] Koufi, L., *et al.*, A Numerical Study of Indoor Air Quality in a Ventilated Room Using Different Strategies of Ventilation, *Mechanics and Industry*, 18 (2017), 2, pp. 1-9
- [24] Younsi, Z., *et al.*, Numerical Study of the Effects of Ventilated Cavities Outlet Location on Thermal Comfort and Air Quality, *Int. J. of Numerical for Methods for Heat and Fluid-Flow*, 29 (2019), 11, pp. 4462-4483
- [25] Ampofo, F., Karayiannis, T. G., Experimental Benchmark Data for Turbulent Natural-Convection in an Air-Filled Square Cavity, *International Journal of Heat and Mass Transfer*, 46 (2003), 19, pp. 3551-3572
- [26] Cao, G., *et al.*, A Review of the Performance of Different Ventilation and Air-Flow Distribution Systems in Buildings, *Building and Environment*, 73 (2014), Mar., pp. 171-186
- [27] Awbi, H. B., *Ventilation of Buildings*, 2nd ed., Routledge, New York, USA, 2003