*International Journal of Energetica (IJECA)*
*https://www.ijeca.info*
*ISSN: 2543-3717 Volume 6. Issue 1. 2021* *Page 01-06*

# Simulation of Night Cooling Through Natural Cross Ventilation using ANSYS (Fluent)

Yamina Harnane[1,2*], Sihem Bouzid[1,4], Sonia Berkane[3], Abdelhafid Brima[2,3] Abdelmadjid Kaddour[5]

[1]*University of Oum El Bouaghi, ALGERIA*
[2]*Laboratory of Mechanical Engineering (LGM), University of Biskra, ALGERIA*
[3]*University of Batna 2, ALGERIA*
[4]*Laboratory of advanced design ad modeling of mechanical and thermo-fluid's system (LCMASMTF), University of Oum El Bouaghi , ALGERIA*
[5] *Unité de Recherche Appliquée en Energies Renouvelables, URAER, Centre de Développement des Energies Renouvelables, CDER, 47133, Ghardaïa, ALGERIA*

Email* : harnaney@gmail.com

**Abstract** – *In this study, we carried out a numerical simulation using CFD code "Fluent 14.0" to quantify night ventilation due to convective and radiative phenomena with well-defined boundary conditions. The configuration is an open square cavity. Top & bottom walls are adiabatic, however, vertical walls represent the left/interior wall and right/external wall provided with a top and a bottom opening, at $T_{cold}$ & $T_{hot}$ temperatures, respectively. The computational domain is two-dimensional with open boundary conditions of the local Bernoulli type. The fluid is incompressible with Boussinesq's approximation and flow regime is stationary turbulent with k-ε RNG model on a 200 * 240 mesh refined near the walls, $Ra_H = 1.43 \times 10^{10}$ (ΔT = 10 °C). The obtained results allowed flow dynamics & thermal characterization as well as cooling integral quantities calculation. Introduction of surface emissivity influences heat transfer via active walls and increases (decreases) the lower (upper) passive wall temperature, while no effect was noted on the dynamics.*

***Keywords**: Thermal comfort, simulation, ventilation, open cavity, natural convection*
Received: 24/04/2021 – Accepted: 29/05/2021

## I. Introduction

A comfortable and energy efficient building is well airtight, well thermally insulated and hygienically well ventilated. Humans spend between 80% and 90% of their time in a closed indoor space and breathe indoor air that is often more polluted than outdoor air. Ventilation allows renewing of the impure air with fresh and healthy air [1], it also allows to cool the buildings internal mass and participate in the body thermal comfort via extracting heat from it by convection and evaporation of sweat. We can find in the literature many more or less recent works in which several experimental models and numerical simulations have been carried-out to evaluate the ventilated cavities performance and their advantages in cooling [2]. In these cavities, air movements are often caused through combined action of pressure gradients induced by wind and/or by thermal draft. Generally, the natural ventilation rate is high when there is a large temperature gradient between indoors and outdoors and/or when air is blowing strongly [3-4]. However, one condition remains to be met, the openings must be made so that progression of circulating air is not impeded [5].

Many numerical studies were carried-out to ensure comfortable indoor conditions while removing effectively heat and contaminant. For example, studies who carried-out researches either on the outlet influence on ventilation performance, air flow field and air mean age distributions, or on the size and location of the thermal contaminant source and its influence on cooling efficiency [7, 8]. Therefore, where the building has several openings placed on opposite facades, the ventilation which takes place is of cross type.

On the other hand, night cross ventilation can be used to release the heat stored during the day in the building envelope. This cross-ventilation mechanism is a good means used in buildings passive cooling to maintain thermal comfort conditions. In hot climates, buildings passive cooling is a proven solution, which is organized around four principles: minimizing internal and external heat input, bringing inertia to the building and ensuring good ventilation to promote convective exchanges. To this purpose, we present this study to describe thermo-convective transfers (evaluation of mass flow rates and heat transfers) used in open cavities (rooms with cross ventilation) in hot climates to improve our knowledge on natural ventilation. Dynamics and thermics flow control in ventilated cavities are difficult because of the intervention of different dynamic and physical parameters. In this optic, we present modeling via a CFD code of natural ventilation in this kind of configuration.

## II. Geometric configuration and boundary conditions

The geometry is a ventilated square cavity of $2.5 \times 2.5\, m^2$. Top and bottom walls (ceiling and floor) are adiabatic while vertical walls (active) represent the left wall (interior wall) at $T_{cold}$ temperature, provided with a 0.3 m top opening (outlet) and right wall (external wall) at $T_{hot}$ temperature, provided with a 0.6 m bottom opening, respectively. The door and the window are closed, only transoms (openings) located at door's top and window's bottom are open, Figure 1.

Our study is simplified to a two-dimensional domain and calculation is limited to the internal cavity. Radiation in this part is negligible, while open boundary conditions are free and of local-Bernoulli type. The air velocity at openings inlet is unknown, inlet & outlet conditions for this kind of model are of "pressure-inlet" $P_{in}(x = 2.5, y_{in}) = -1/2\, u^2(x = 2.5, y_{in})$ and "pressure-outlet" $P_{out}(x = 0, y_{out}) = 0$ type, respectively. Inlet turbulence intensity is equal to 5% [1] and with a hydraulic diameter $\mathrm{D_H} = 2 \times 0.6 = 1.2$ m. Outside air temperature is $T_0 = 298.15\, K$ and vertical walls temperatures are fixed and constant. $T_{cold} = T_o$ and $T_{hot} = T_o + \Delta T$

When radiation is considered, both opposite isothermal walls (active walls, ε = 0.15) and other passive walls (adiabatic: floor and ceiling) have an emissivity of [0.1; 0.6; 0.9]. The air inside the cavity is a semi-transparent medium with an absorption coefficient of $\alpha = 0.1\, m^{-1}$ . The fluid is incompressible with the Boussinesq approximation and the regime is turbulent and stationary.

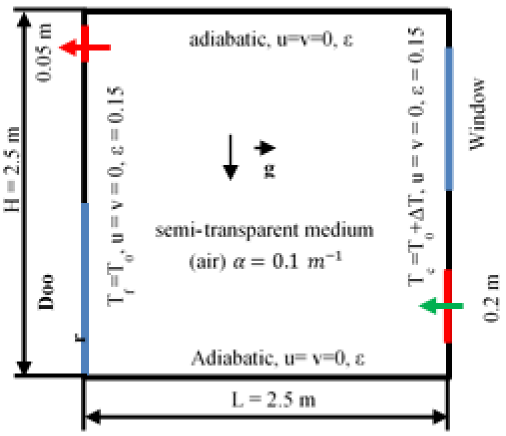



Figure 1. Ventilated cavity geometry with boundary conditions

## III. Mathematical formulation and governing equations

*Continuity equation:*

$$\frac{\partial \rho}{\partial t} + \frac{\partial u_i}{\partial x_i} = 0 \qquad 1 \le i, j \le 3 \qquad (1)$$

*Momentum equation:*

$$\frac{\partial \rho u_i}{\partial t} + u_j \frac{\partial(\rho u_i)}{\partial x_j} + \rho u_i \frac{\partial u_j}{\partial x_j} = \left(-\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_i}\right) + f_i \qquad (2)$$

Where $\tau_{ij} = 2\mu\left(\frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right)$

*Energy equation:*

$$\rho\left(\frac{\partial h}{\partial t} + u_j \frac{\partial h}{\partial x_j}\right) = \tau_{ij}\frac{\partial u_i}{\partial x_j} + \left(\frac{\partial p}{\partial t} + u_j \frac{\partial p}{\partial x_j}\right) - \frac{\partial q_j}{\partial x_j} + \Phi \qquad (3)$$

*Equation of state:*

$$\rho = \rho(p, T, x_i) \qquad (4)$$

The Boussinesq hypothesis assumes that fluid density in terms of volume forces varies linearly with temperature, which leads to the following relationship:

$$\rho = \rho_0[1 - \beta(T - T_0)] \qquad (5)$$

k-ε RNG model, developed by Yakhot V. and Orszag S.A [2], is based on a theory known as "renormalization". The constants relating to this model are as follows:

$C_\mu = 0.0845$ ; $C_{\varepsilon 1} = 1.42$ ; $C_{\varepsilon 2} = 1.68$ ; $\sigma_T = 0.9$ ; $\sigma_k = 1.0$ ; $\sigma_\varepsilon = 1.3$

For thermal radiation modeling, we have chosen the discrete ordinates model DO [3-4].

Radiative transfer equation (ETR):

$$\frac{dI(\vec{r},\vec{s})}{ds} + (\alpha + \sigma_s) I(\vec{r},\vec{s}) = \alpha n^2 \frac{\sigma T^4}{\pi} +$$

$$\frac{\sigma_s}{4\pi}\int_0^{4\pi} I(\vec{r},\vec{s})\Phi\left(\vec{s},\vec{s'}\right)d\Omega' \tag{6}$$

### III.1. Mean age of air (MAA)

The thermal comfort indices which used in this study can be calculated as following:

The mean age of air can be calculated from the following transport equation:

$$\frac{\partial}{\partial x}\left(\rho u_i MAA - \left(2.88\rho \times 10^{-5} + \frac{\mu_{eff}}{0.7}\right)\frac{\partial MAA}{\partial x_i}\right) = S \tag{7}$$

Where S is the source term depending on the air density. The MAA is not directly available from fluent so, it is programmed and calculated as user-defined scalars. [9-13].

### III.2. Effective draft temperature (EDT)

Effective draft temperature (EDT) is one of the first thermal indexes. It combines temperature and air velocity. EDT values between -1.7 and 1.1 (-1.7 < EDT < 1.1) characterize thermal comfort while EDT values outside this range, represent thermal discomfort zone [14-16]. Values less than -1.7 represent cool sensation while values above 1.1 represent warm sensation. According to [15], EDT is defined as:

$$EDT = (T_x - T_m) - 8(V_x - 0.15) \tag{8}$$

## IV. Resolution procedure

For this study, we used the k-ε RNG model with Enhanced-wall function on a $200 \times 240$ mesh (48000 cells) refined near the walls. Equations resolution is carried-out via "SIMPLE" algorithm by adopting, from one hand, the "2nd Order" scheme for pressure and diffusive terms, and on the other hand, the 2nd Order "UPWIND" scheme for convective terms. Rayleigh number is important $Ra_H = 1.43 \times 10^{10}$ (ΔT=10 °C). The main physical quantities, studied within buildings night cooling framework by natural ventilation, i.e. ventilation flow rate, the heat exchanges and the cooling power, are determined by the following formulas: [5].

- Cross ventilation flow [m$^3$/h]

$$q_v = 3600\int \vec{V}.\vec{n}.dS_{inlet} \tag{9}$$

- Average Nusselt numbers at the walls
- Air renewal rate [vol/h]

$$\eta = q_v/volume \tag{10}$$

- Outlet fluid average temperature $\theta_{m\,\mathrm{avg}}$ [-]

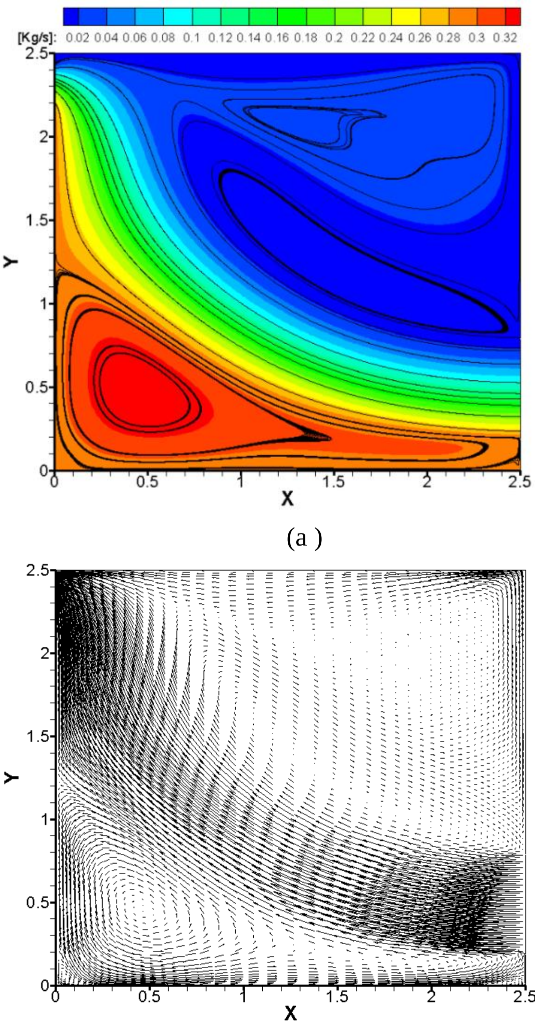
$$\theta_m = \frac{\int_{outlet}\theta\vec{V}.\vec{n}.dS}{\int_{outlet}\vec{V}.\vec{n}.dS} \tag{11}$$

- Cooling capacity [W]

$$Q = \rho q_v C_p \theta_m \Delta T \tag{12}$$

## V. Results and interpretation

Two main cells will appear extending horizontally along the floor below the jet and along the ceiling. The third cell is located between the jet and the second cell; Figure 2-a. The resulting velocity field, Figure 2-b indicates the flow path and the air jet pace from inlet to outlet. The vectors positive direction shows the recirculation areas. On the ceiling, we can clearly see the appearance of dynamic boundary layers and the maximum values of the velocity are located in the jet.




(b)

Figure 2. (a) Current function; (b) Velocity field

### *V.1. Mean age of air*

MAA values reflect the supply air flow characteristics and can, therefore, be adopted to evaluate supply air distributions. As seen in Figure 3, there is a large region with high MAA values, up to 90 s in the upper right part zone of the cavity, indicating that the supply air had a little effect on the air movement in this region.

MAA value in the region near the floor is significantly lower, between 30 and 50 s. In the jet, MAA value is much lower, suggesting that it took less time to deliver the supply air to this region than to other ones.

MAA lower values don't allow the air to exchange the heat with the walls. This means that the air is extracted from the cavity with low temperature. Hence, the ventilation effectiveness is high.

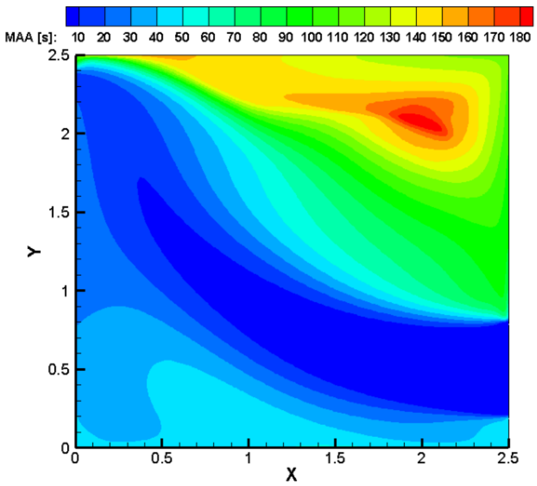



Figure 3. MAA distribution inside the ventilated cavity

### *V.2. EDT index and thermal field*

The thermal field indicates that a jet of cold air goes from the bottom opening to the top outlet, it divides the flow into two main streams. The first one is a cold stream that crawls along the floor to the opposite wall, it rises to the exit. The second is a jet heated by the right wall and rises along it to reach the exit but remaining stuck to the ceiling. The heart of the cavity is well cooled; Figure 4-a.

In order to predict thermal comfort zones, the EDT index was calculated. From formula (8), we see that the effects of only two parameters of air, namely temperature and velocity, which were used to form this index. The EDT shows high sensitivity of temperature and air velocity over the thermal comfort zone. The EDT index greater than +1.1 indicates a hot discomfort zone and when EDT is less than -1.7, we speak of a cold discomfort zone.

In Figure 4-b, we notice a zone of cold discomfort created from the entrance which extends along the jet to the exit. The comfort zone does not exceed 0.5 m elevation. Also, due to the input jet effect, a hot discomfort zone is created at the right upper part of the cavity. As a result, the thermal comfort zone covers the lower area under the jet.

The effect of the inlet jet on the thermal discomfort zone near the entrance is very pronounced and extends from the entrance to the exit of the cavity. This case illustrates that the whole body is inside the comfort zone on the left area of the cavity.

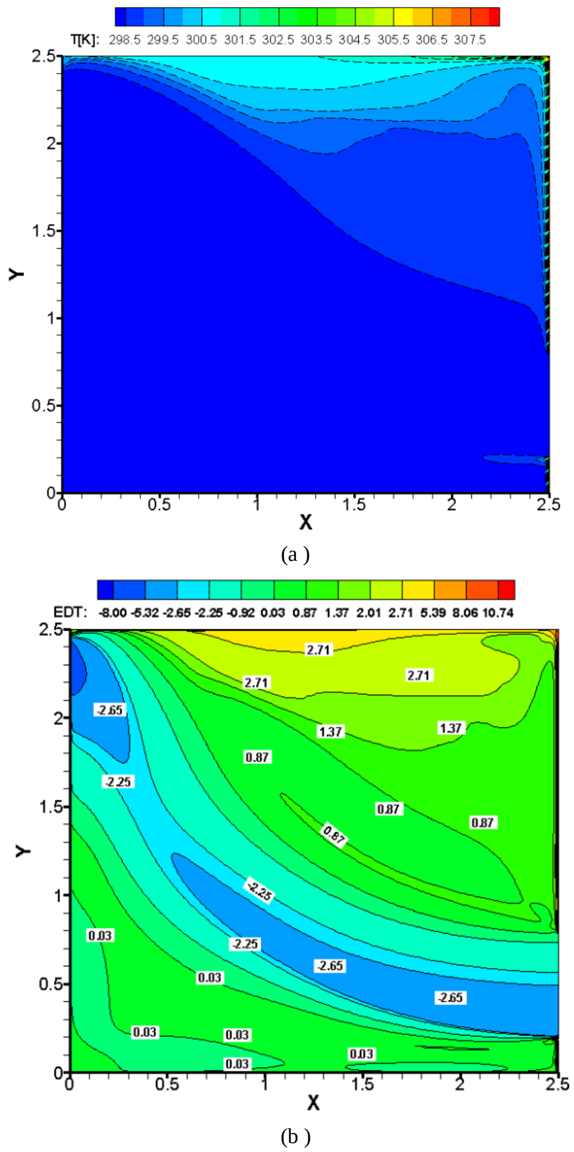



Figure 4. (a) Thermal field; (b) EDT distribution inside the ventilated cavity

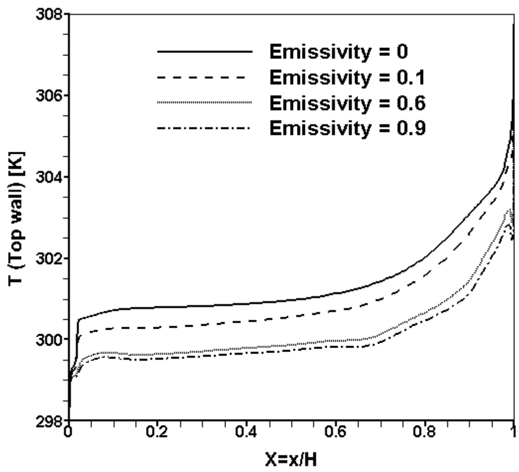
Table 1 shows the results obtained for the integral quantities (calculated by Excel). We find that the air renewal rate is very high which is very interesting for night cooling.

Table 1. Integral quantities

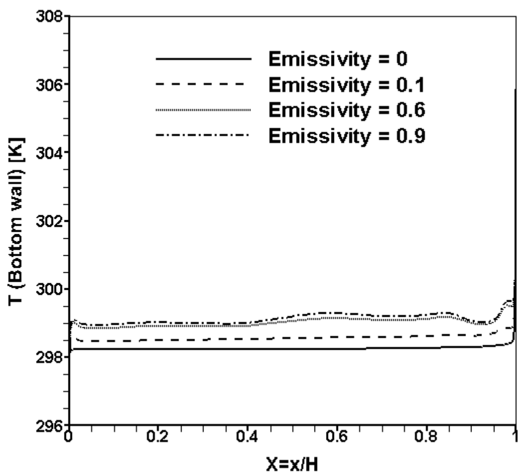
| $Ra_H$ | $\theta_m$ | $q_v$ | Q |
|---|---|---|---|
| $1.43*10^{10}$ | 0.215 | 329.65 | 235.37 |

| $Ra_H$ | η | $<Nu>^f$ | $<Nu>^c$ |
|---|---|---|---|
| $1.43*10^{10}$ | 52.74 | -2.67 | 114 |

Surface convection-radiation coupling occurs only via adaptability condition at passive walls. Temperature distribution along the cavity top wall (adiabatic wall) shows an almost homogeneous average temperature (≈T = 300.92K) with a slight decrease. In Figure 4, we note that temperature decreases along with passive walls $\boldsymbol{\varepsilon_p}$ emissivity increasing. Consequently, wall radiation would reduce the top wall temperature allowing heat exchanges with the other walls. For the cavity lower wall, the phenomenon is reversed. This phenomenon has already been described in scientific study [17].




(a )




(b)

Figure 4. Evolution of passive walls temperature as a function of emissivity, (a) top wall, (b) bottom wall.

Total Nusselt number is given via the formula:

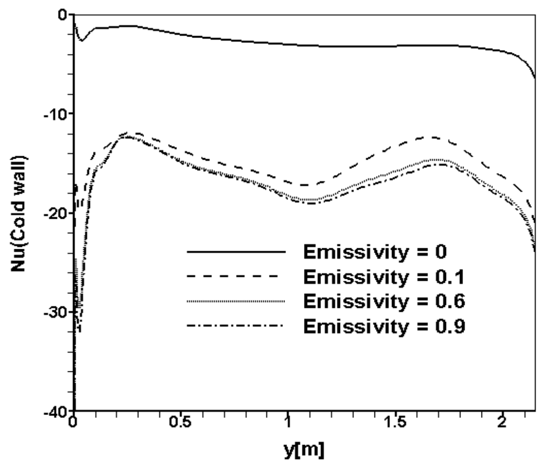
$$Nu_g = Nu_{conv} + Nu_{rad}$$

Increasing $\boldsymbol{\varepsilon_p}$ emissivity increases all the convective exchanges at the active walls, as shown in Figure 5. Surface radiation contribution at the active walls is important, it is about 84%, compared to the hot right wall which is about 24%, "Table 2". This can be explained through the air jet effect impacting the cold wall.

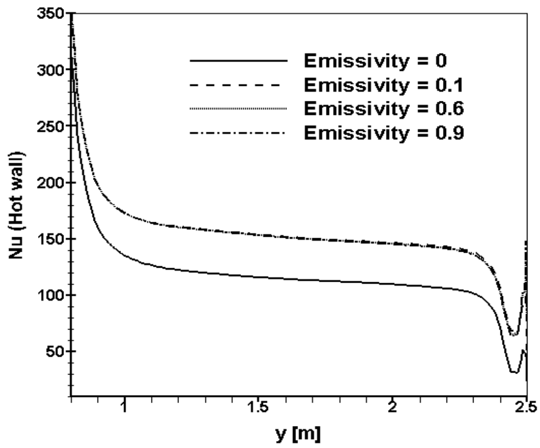
Table 2. Total Nusselt numbers at the walls (Emissivity increasing effect)

| | $<Nu>^{cold}$ | $<Nu>^{hot}$ | $<Nu_{rad}>^{hot}$ |
|---|---|---|---|
| ε=0 | 2.677 | 113.99 | 0 |
| ε=0.1 | 15.124 | 151.416 | 37.43 |
| ε=0.6 | 17.662 | 151.136 | 37.15 |
| ε=0.9 | 18.263 | 151.09 | 37.1 |

| | $<Nu_{rad}>^{cold}$ | $(Nu_{rad}/Nu_g)^{cold}$ | $(Nu_{rad}/Nu_g)^{hot}$ |
|---|---|---|---|
| ε=0 | 0 | ----- | ----- |
| ε=0.1 | 12.48 | 82% | 23% |
| ε=0.6 | 14.98 | 84.84% | 24.6% |
| ε=0.9 | 15.58 | 85.34% | 24.5% |




(a )




(b)

Figure 5: Evolution of active walls Nusselt number as a function of emissivity, (a) cold wall, (b) hot wall.

## VI. Conclusion

Turbulent flow in the ventilated cavity was well simulated through CFD calculation code "Fluent". Our goal is to study flow's dynamics and thermics for night cooling of a room similar to a ventilated cavity. We evaluated energy performances and integral quantities (η, θm, $q_v$, Q) for this kind of configuration in presence of radiation and without radiation. The main conclusions are as follows:

The configuration with openings set on opposite walls promotes night cooling with a high air exchange rate and the surface radiation intervening through the walls has an effect only on walls temperatures and Nusselts. Its effect on the flow's thermics and dynamics is negligible.

## Nomenclature

| | | | |
|---|---|---|---|
| $D_H$ | hydraulic diameter, m | *Indices* | |
| $H$ | height, m | *conv* | convective |
| $L$ | width, m | *g* | global |
| $l$ | wall thickness, m | *m* | mean |
| $P$ | pressure, Pa | *in* | inlet |
| $T$ | temperature, K | *out* | outlet |
| $T_x$ | local air temperature, °C | *rad* | radiative |
| $T_m$ | mean temperature, °C | *x* | local |
| $V_x$ | local air velocity, m.s$^{-1}$ | | |
| $\theta$ | dimensionless temperature $= (T - T_0)/\Delta T$ | | |

## References


[1] FLUENT User's Guide, Fluent Inc., Release 14.0, November 2011

[2] V. Yakhot, S. A. Orszag, "Renormalization group analysis of turbulence I, Basic theory, Journal of Scientific Computing, Vol. 1, No 1, 1986, pp. 3-51.

[3] A. Ibrahim, D. Saury, D. Lemonnier, "Coupling of turbulent natural convection with radiation in an air-filled differentially-heated cavity at Ra = 1.5*10$^9$", Computers & Fluids, Vol. 88, 2013, pp.115–125.

[4] W. Fiveland, "Discrete-Ordinates Solutions of the Radiative Transport Equation for Rectangular Enclosures", Journal of Heat Transfer, Vol. 106, 1984, pp. 699–706.

[5] B. Brangeon,, "Contribution à l'étude numérique de la ventilation naturelle dans des cavités ouvertes par la simulation des grandes échelles. Application au rafraîchissement passif des bâtiments". Thèse de Doctorat Université de la Réunion, 2012.

[6] M. Ning, S. Mengyin, P. Dongmei, D. Shiming, Computational fluid dynamics (CFD) modeling of air flow field, mean age of air and CO2 distributions inside a bedroom with different heights of conditioned air supply outlet. Applied Energy, Vol. 164, 2016 pp. 906-916.

[7] Z. Younsi, L. Koufi, H. Naji, Numerical study of the effects of ventilated cavities outlet location on thermal comfort and air quality. International Journal of Numerical Methods for Heat & Fluid Flow, Vol. 29. No 11, 2019 pp. 4462-4483.

[8] N. Gupta, A. K. Nayak, S. Malik, Conjugate heat and species transport in an air-filled ventilated enclosure with a thermo-contaminated block. IJHMT, Vol. 117. 2018, pp. 388-411

[9] F. R. Menter, Two-equation eddy-viscosity turbulence models for engineering applications. AIAA J, Vol. 32. No 8, 1994 pp. 1598–1605.

[10] C. Hu. T. Kurabuch, M. Ohba. Numerical study of cross-ventilation using two equation RANS turbulent models. Int J Ventilation, Vol.4. No 2, 2005, pp.123–132.

[11] A. Stamou, I. Katsiris. Verification of a CFD model for indoor airflow and heat transfer. Build Environment. Vol. 41, 2006, pp. 1171–1181.

[12] G. D. Raithby, E. H. Chui. A finite-volume method for predicting a radiant heat transfer in enclosures with participating media. J Heat Transfer. Vol. 112. No 2, 1990, pp. 415-423.

[13] V. Chanteloup, P. S. Mirade. "Computational fluid dynamics (CFD) modelling of local mean age of air distribution in forced-ventilation food plants". J Food Engineering. Vol. 90, No 1,2009, pp. 90-103.

**[14]** J. Rydberg J and P. Norback, "Air Distribution and Draft", ASHRAE Transactions, 1949, pp. 55-225.

[15] A. Koestel and G. L. Tuve. "Performance and Evaluation of Room Air Distribution Systems", ASHRAE Transactions, 1955, pp. 61-533

[16] A. F. Alfahaid, Effects of ventilation on human thermal comfort in rooms, Ph. D Thesis, old dominion university, Norfolk, Virginia 2000.

[17] H. Wang, S. Xin, P Le Quéré, "Etude numérique du couplage de la convection naturelle avec le rayonnement de surfaces en cavité carrée remplie d'air ". C.R. académie des Sciences. Vol. 334, 2006, pp 48-57.