**Islamic University Journal of Applied Sciences (IUJAS)**
https://journals.iu.edu.sa/jesc

Volume VIII, Issue I, July 2026, Pages 218-229

# Numerical study of air flow and temperature distribution in a room equipped with a cooling ceiling

Rim Ben Khalifa [1], Mohammed Awdah B Alshehre [2], Amamou Amani [1,3] and Nejla Mahjoub Said [2,*]

[1] LGM, National Engineering School of Monastir, 5000 Monastir, University of Monastir, Tunisia, rimbenkalifa@yahoo.fr

[2] Department of Physics, College of Science, King Khalid University, Abha 61413, Saudi Arabia, 445818821@kku.edu.sa, nalmahjoub@kku.edu.sa

[3] National Engineering School of Gasfa, University of Gafsa, 2112 Gafsa, Tunisia, amani.amamou@yahoo.fr

*Corresponding author: (Nejla Mahjoub Said), *Email Address*: nalmahjoub@kku.edu.sa

## Abstract

The purpose of this paper is to simulate the most important problems in thermal comfort requirements, mainly a cooling ceiling system Using the RANS approach. A field flow model based on computational fluid dynamics was used to simulate the heat transfer of a room with the ceiling cooling system. The simulation is carried out using the Shear Stress Transport k-ω model. The main objective is to evaluate the heat transfer of a room equipped with a cooling ceiling. Firstly, the study describes the thermal performance of the cooling ceiling. Then, the effect of the water velocity on the performance of the floor is presented. Finally, the cooling ceiling system in a room is analyzed in terms of hydrodynamic and thermal performance for different Reynolds number. In addition, the results indicate that the cooling capacity increases nearly linearly with the temperature difference between the air-room and the cooling ceiling system.

**Keywords:** Dynamics (CFD); RANS turbulence modelling; Thermal comfort; Heat transfer and Reynolds number effects.

https://doi.org/10.63070/jesc.2026.024
Received 24 December 2025; Revised 12 April 2026; Accepted 16 April 2026.
Available online 20 April 2026.
Published by Islamic University of Madinah on behalf of *Islamic University Journal of Applied Sciences*.
This is an open access article published under the Creative Commons Attribution 4.0 International license (CC BY 4.0):https://creativecommons.org/licenses/by/4.0/

## 1. Introduction

Regarding changes in the climate and fossil fuel limitations, the growing demand of energy all over the world is becoming a serious problem. New and renewable energy sources seem to be the solution. The building sector is one of the largest consumers of energy with a large amount directly related to the heating and cooling of buildings.

As is well known in buildings, the presence of people, equipment, sunlight, or lighting generates a very large amount of heat that makes cooled-air temperature regulation systems insufficient. Many previous studies present the various modes and processes of heat transfer. Increasing the energy efficiency of the building is an alternative to reduce energy consumption while improving thermal comfort. The “Future of Cool”, where new technologies like cooling ceiling system should continue to grow in popularity for many buildings. This new technology to cool air uses much less energy and with a higher comfortable degree in the field of air-conditioning. Currently, ceiling cooling systems are increasingly in use. They are used for both cooling in the summer period and heating in cold periods.

Several researchers have studied the cooling performance under the influence of various parameters. The various modes and processes of heat transfer are discussed by Graziani et al. [1] and Xia et al. [2].

They made the transfer of heat to or from a room equipped with a cooling ceiling by two-mode of transfers: convection and radiation modes. In an experimental study Chicote et al. [3] studied the cooling capacity and heat transfer coefficients. In a test room equipped with a cooled radiant ceiling, they found that radiant and convective phenomena must be examined separately. They also determined that the radiant heat transfer coefficient attains lower values due to the influence of the ceiling surface emissivity. The total cooling capacity of the radiant ceiling can be determined by taking into consideration both radiant and convective heat fluxes.

The cooling capacity and heat transfer coefficients were the objectives of the numerical study of Refet [4]. With this study, the authors gave a new equation related to the convective heat transfer coefficient, heat transfer coefficient and the ratio of heat transfer coefficients. This equation depends only on the temperature difference between the indoor air and ceiling surface.

With the study of Jeong et al. [5], they made A comparison between the pilot DOAS/ radiant panel cooling system and a conventional all-air VAV system with air-side economizer. They found that the pilot DOAS/ radiant panel cooling system using more pumping energy. Furthermore, they showed that when the ceiling coverage ratio of suspended panels decreases, panel-cooling capacity increases. Chiang, et al [6] examined with a full-scale experiment a radiant cooling ceiling system installed in an

office. The results were compared with the values from the CFD simulation. Many parameters were analyzed such as air temperature from the diffuser, surface temperature, and the area of the cooling panels. This study improves indoor thermal comfort and energy efficiency using a radiant cooling ceiling system.

A series of numerical research on the ceiling cooling system were carried out. We can find the numerical simulation of Xie et al. [7]. In this computational fluid dynamics (CFD) simulation study, the author examined the heat transfer of water flow in the capillary of ceiling radiant cooling panel. Six influencing factors such as chilled water inlet parameters, conditions of gypsum plaster and capillary mats structural parameters were examined to obtain heat transfer performance. Yin et al [8] examined with a numerical simulation, a metal ceiling radiant cooling panels with a serpentine tube arrangement (CRCP-s). Further in this study, the research investigated the influences of the mean panel temperature, outlet water temperature and cooling capacity. The results show that all the parameters have a significant effect on the cooling capacity.

Ashrae [9] conducted experiments to examine the performance of using a cooling ceiling panel to transfer heat. This research refers to the effect of chilled water temperature, supplied chilled water flow rate, and the cooling performance of the panel. The experiment showed that supplied chilled water temperature has a direct influence on the surface temperature of the panel, the room temperature, and the heat transfer rate of the panel.

Causone et al. [10] evaluated experimentally the characters of heat transfer in the case of radiant cooling ceiling equipped in a room. It is remarked that several parameters could be affected the heat transfer coefficient: surface temperature distributions, internal gains, air movements. Experimental data emphasizes the fundamental role of the reference temperature to determine the heat transfer coefficients. The reference temperature is defined as the average unheated surface temperature (AUST) calculated considering the view factors between surfaces

In these terms, the study of Su et al. [11] analyzed the concrete ceiling radiant cooling panel. The temperature distribution of the concrete panel is represented under different conditions: water temperatures, distances of tubes, and water flow rate.

Also, the authors found that the large thermal inertia of the concrete panel caused the temperatures of the panel and indoor air to drop slowly. However, it reduced the adverse impact of supply water temperature fluctuation. This previous work is aimed at developing practical models to estimate the cooling output on indoor thermal comfort.

Karathanassis et al [12] investigated, with a numerical simulation, the flow and temperature fields in the case of ceiling cooling systems. In another study, Menter [13] introduced the Shear Stress Transport (SST) k-ω model. They, also, evaluated the prediction of the turbulent flow and heat transfer. The flow phenomena emerging inside the tubes were examined and thoroughly discussed. Sikula et al [14] analyzed the dynamic behavior of the temperature field of the cooling ceiling. The simulation was carried out using the Fluent 6.3 computer programs. The model of turbulence chosen was RNG k-ε to quantify in detail the cooling ceiling capacity and the ceiling surface temperatures for different Reynolds Number.

Many parameters are examined in different study like Reynolds Number, the dimension of the room

Also, we found a numerical and experimental study are focused to analyze the effect of Water velocity. Fredriksson and al. [15] found different phases of velocity: after an initial acceleration phase the velocity is almost constant. Then there is a slow decay in velocity with the distance from the chilled beam. This is attributed to the finite length of the chilled beam, which gives rise to entrainment of air at the ends of the plume. A CFD simulations is realized with the aim to analyse the airflow pattern and the velocity fields inside the test room in the study of Catalina [16]. It was shown in their work that a local discomfort at feet/ankle zone was observed but with good values of air velocity for the rest of the test room.

This paper reports a numerical study of a cooling ceiling installed in a test room. Ansys Fluent is a powerful CFD software that is powerful and up to date in all simulation fields related to fluid mechanics and heat transfer. The Shear Stress Transport (SST) k-ω model already implemented to predict the mean flow field and the heat transfer in the boundary layer. The mean temperatures, the behavior of the global flow generated by a cooling ceiling are presented. Additionally, the effects of the velocity of water on the thermal performance are examined. Finally, the flow and temperature fields in a room equipped with a cooling ceiling are analyzed.

## 2 Numerical simulations

### 2.1 Governing equations

Stokes equations (RANS equations), based on the Reynolds decomposition, are numerically solved. For such flows, the following assumptions are made in this numerical simulation: steady and incompressible fluid flow. Based on these assumptions, the governing equations including the continuity (1), the momentum (2) and the energy equation (3), are considered as follows:

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

$$\frac{\partial}{\partial x_k}(\rho u_k u_i) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_k}\left[\mu\left(\frac{\partial u_i}{\partial x_k} + \frac{\partial u_k}{\partial x_i}\right) - \rho\overline{u_i' u_k'}\right] \quad (2)$$

$$\frac{\partial}{\partial x_k}(\rho u_k T) = \frac{\partial}{\partial x_k}\left(\frac{\mu}{Pr}\frac{\partial T}{\partial x_k} - \rho\overline{u_k' T'}\right) \quad (3)$$

Where i, k =1, 2, 3 referring to x, y and z respectively. $u_i$ and $x_i$ are the mean velocity and the coordinate in the i direction, respectively. P represents the pressure and ρ is the density. μ and Pr are the dynamic viscosity and the Prandtl number, respectively. T designates the temperature. The Reynolds Averaged Navier-Stokes (RANS) equations formulated above contain the Reynolds stresses terms $\overline{u_i' u_k'}$ and the turbulent heat flux $\overline{u_k' T'}$. The Shear Stress Transport (SST) k-ω model introduced by Graziani et al. [1] is used for the calculation of these terms.

### 2.2. Flow configuration and boundary conditions

The main dimensions of the test room with a cooling ceiling are given in Figure 1. In order to examine the cooling performance, the study was conducted in an office room (1.2m × 0.55 m × 0.6 m). It is equipped with a hydronic ceiling composed of six serpentine tubes. Airflow will enter the room through a door and check the measurements through a window. The rest of the domain was considered an adiabatic wall.

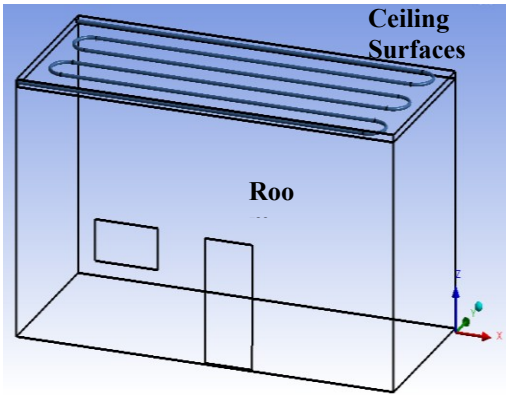



Figure 1. The main dimensions of Room with Cooling Ceiling System

The ceiling cooling panel is installed in the room. The cooling device comprises gypsum serpentine tubes thermally bonded to rectangular aluminum. The simulation was performed under these conditions: the surface temperature of all walls was set ambient temperature ($T_a$=24°C) and the cooling water temperature "$T_w$" in the tubes of the ceiling was set to $T_w$=10°C. The air temperature of the chamber was kept constant at 24 °C. The dimension of the main numerical domain is listed in Table 1. The cooling ceiling configuration is inspired from the flow configuration of Yin et al. [8]. The dimensions of the radiant panel comprise three distinct layers: an insulation layer, a water pipe layer, and a surface layer. (Figure 1).

Table 1: Main dimensions and materials

| Items | Values |
|---|---|
| Tube outside diameter (D) | 0.009 m |
| Number of tubes (N) | 6 |
| Panel thickness (spanwise direction ($z$)) | 0.015m |
| Panel material | Gypsum |
| Length (axial direction ($x$)) | 1.2 m |
| Width of ceiling surface (vertical direction ($y$)) | 0.55m |
| Velocity air ($V_a$) | 0.3 $m^2/s$ |
| width of Room ((axial direction (z)) | 0.6m |

## 3. Numerical method

The resolution of the governing equations with appropriate boundary conditions is carried out, by using Ansys Fluent [17]. The numerical code uses the finite volume method; the solution of equations being based on the algorithm SIMPLER proposed by Patankar [18]. For the algebraic equations obtained, we use an implicit scheme.

The convergence absolute criterion is set to a residual value less than $10^{-5}$ for the continuity and velocity component equations and afterward, the energy equation.

The computational domains for both two parts "room and cooling ceiling" was discretized using unstructured grids. A sufficiently fine grid was closely positioned adjacent to the solid ceiling panel where prevails a very high gradient of different variables.

## 4. Results and discussion

### 4.1 Analysis the total thermal performance of the cooling ceiling

#### *4.1.1 Flow Field*

Figure 2 gives an example of the evolution of the surface temperature on the lower face of the ceiling.

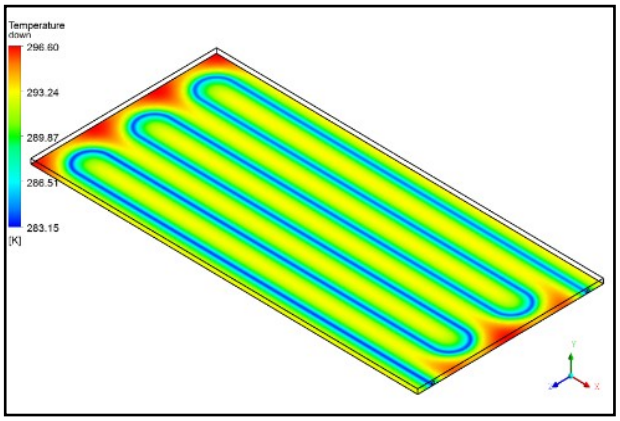



Figure 2. Scheme of temperature fields of ceiling surface

Tests were conducted for a Reynolds number of 2077, based on the diameter of pipe "D" and the water velocity "$V_w$".). The Reynolds number for pipe flow is defined as $Re = V_w \times D/\nu$.

As expected, the lowest temperatures are below the cooling pipes. It is obvious that the surface temperature distributions could be found a similar trend depicted in literature [7].

A large temperature gradient occurs between water fluid and gypsum ceiling, in all cases, to be driven by convection from the adjacent walls. It can be seen from this figure the values were uniformly located at the panel surface. One can also notice ceiling surface temperatures near the outdoor walls that are significantly higher - around 24 C.

The temperature fields in a longitudinal plane x–y for three planes (x = 0.4 m, x = 0.6 m and x = 1.15m) is shown in figure 3. This figure shows streamlines of temperature, observed for different positions to determine the spreading behavior of the temperature. The water temperature affects the surface temperature of the panel. Cold fluid is deflected toward the outer tube wall, causing the diminution of the ceiling temperature. Downstream of the water pipe, the total temperature distribution is uniform. With a cooling ceiling the temperature of water or its flow rate can be modified so that the surface temperature adapt to the desired conditions.

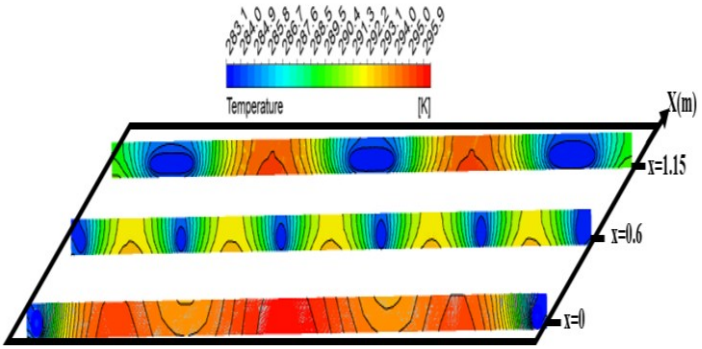



Figure 3. Two-dimensional temperature distribution of the cooling ceiling panel

Figure 4 (a, b) presents the distribution of water velocity in the symmetry plane (y=0 m) and the streamwise velocity profiles of the downstream pipe flow. The cool water flow enters the pipe at a uniform velocity (0.3 m/s). The region beyond the entrance region in which the velocity is changing significantly. As is clear from Figure 4 (a), the velocity of the fluid at the midsection of the pipe must increase to keep the mass flow rate through the pipe constant. As a result, a velocity gradient develops along with the pip. It is clear that the velocity profile appears distorted downstream of the inlet water, at the curved region (Figure 4 b). When the fluid in the pipe is not cooled, in this case, the velocity profile is remains unchanged.

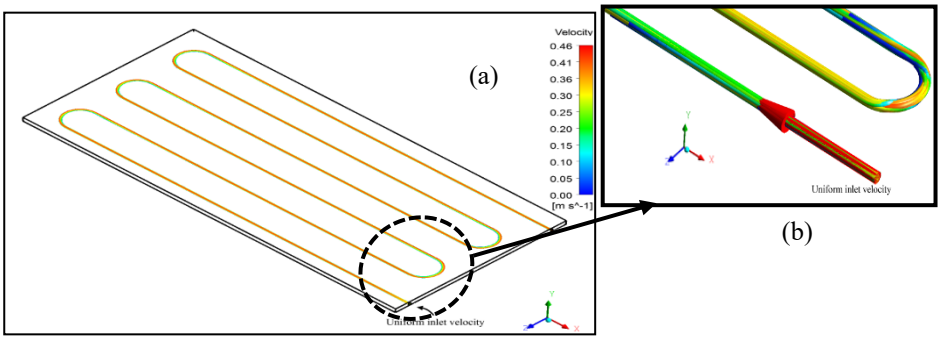



Figure 4. (a) Scheme of velocity fields the symmetry plane, (b) Streamwise velocity profiles of the downstream pipe flow

### *4.1.2 Effect of the Reynolds number*

To further elucidate the global flow of the cooling system, Karathanassis et al [12] examined the flow in the cooling configurations for lower values of the Reynolds number. In our study, the temperature distribution covering a wide range of Reynolds numbers was investigated (Re=692-2770). The Reynolds number for pipe flow is defined as $Re = V_w \times D/\nu$ with ν is the kinematic viscosity of the water.

The distribution of the vertical temperature along the x-direction for different Reynolds numbers is illustrated in figure 5. As shown in this figure, the heat transfer rate depends on the cooling capacity rate of water temperatures. Therefore, the water flow temperature decreases with increasing Reynolds number.

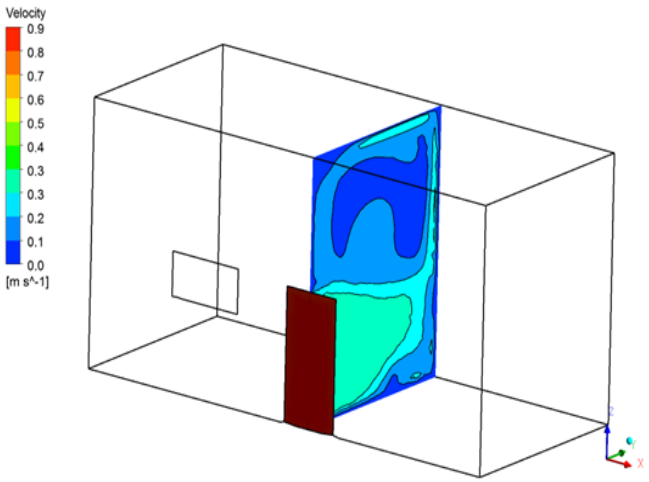



Figure 5. Distributions of the vertical temperature along the x-direction for different Reynolds number

For Reynolds number values greater than 1384, the cooling effect was enhanced, however, with the velocity increasing. The total heat transfer and the cooling capacity reduces with the reduction of

velocity water. Therefore, it can conclude that the velocity water should be large enough for greater cooling capacity.

### 4.2 Analysis of the behaviour of the cooling ceiling in an office room

In our study, the investigation of the heat exchange between the cooling ceiling with the room thermal is presented. Temperature of the surface is an important factor exercising significant influence on the process of heat transfer to air-water mist. The temperature of the surface is shown in Fig. 6 for different Reynolds numbers. The ambient air velocity enters to the room through the door is equal to 0.3 m/s. The vertical temperature fields in the test room in the plane x =0.6 m is examined in this Figure.6.

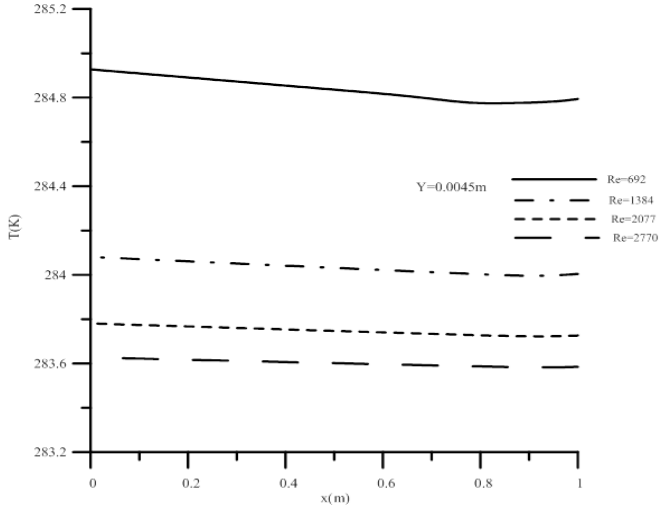



Figure 6. The total temperature fields inside the test room in the plane x=0.6m.

The air movements are occurring in the test room. The warm air enters the room from the door with ambient temperature and is rising to the ceiling. The presence of a cooling ceiling allows the generation of a cold downdraught. The rising air creates an interior low pressure that should pull in large volumes. The heat exchange increases nearly linearly with the temperature difference between the room air and the cooling ceiling system. In this condition, a stable thermal stratification developed inside the room, with warm still air close to the heated ceiling and low convective heat transfer. According with the Figure ,6 Surface temperature decreases with increasing Reynolds number. Because of evaporation of water, surface temperature decreases in the air-water mist with increasing water flow density at constant heat flux. It can be seen so that the higher the heat capacity ratio, the higher the convective heat transfer coefficient. This is because a higher heat capacity ratio leads to a more balanced flow and thus an increase in heat transfer rate.

Figure 7 presents the behavior of the velocity inside the room with the presence of a cooling ceiling system and in the plane x=0.6m. This figure indicates that the room will be divided into a lower region

where the air is like the supply condition and an upper region above the stationary front that is well mixed cold air. It is remarked also that when an airstream strikes the cooling ceiling surface, it tends to spread and follow the surface until the velocity dissipates. Therefore, it is interesting to examine the influence of air velocity because it is seemed to contribute more significantly to the enhancement of cooling panel capacity.

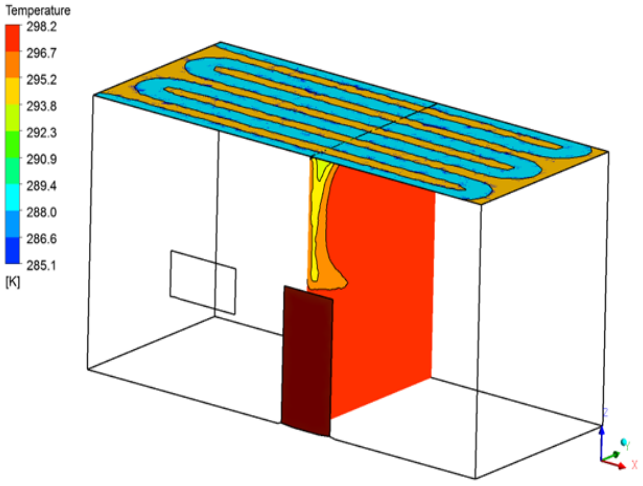



Figure 7. Velocity fields inside the test room in the plane x=0.6m.

## 5. Conclusion

The performance of the cooling ceiling installed in a room has been investigated in this paper using CFD simulations. This examination was conducted numerically employing of the finite volume method together with the Shear Stress Transport (SST) k-ω and a non -uniform mesh system.

Coldwater is spread out along the pipes causing the diminution of the ceiling surface temperature. A higher Reynolds number leads to a lower outlet temperature of water. It was established that an increase in the Reynolds number affects heat transfer. For practical applications, the Reynolds number should be large enough to keep the cooling capacity and the thermal comfort of the room. Secondly, the dynamic characteristics and the thermal behavior in the room with the presence of a cooling system are examined.

The room receives a very even temperature spread between the ceiling and the floor. Significant mixing, indicated by reduced temperature gradients, was evident in the upper part of the room with ceiling cooling. Velocity and Temperature measurements indicated that with greater heat gains the plumes have enough momentum to drive flow across the ceiling surface and down the walls.

**Funding**

The authors declare that this research received no external funding.

**Data Availability Statement**

Not applicable.

**Conflicts of Interest**

The authors declare no conflict of interest.

**References**

[1] S. Graziano, S., Jens, P. Marta, M.S. Assessing energy and thermal comfort of different low-energy cooling concepts for non-residential buildings, Energy Conversion and Management, Volume 76, December 2013, Pages 332-341.

[2] Yizai Xia, Mumma, A. Ceiling Radiant Cooling Panels Employing Heat-Conducting Rails: Deriving the Governing Heat Transfer Equations, ASHRAE Transactions. 2006, Vol. 112 Issue 1, p34-41. 8p.

[3] Andrés-Chicote, M. Tejero-González, A. Velasco-Gómez,E. F. J. Rey-Martínez, E. Experimental study on the cooling capacity of a radiant cooled ceiling system, Energy and Buildings, Volume 54, November 2012, Pages 207-214.

[4] K. Refet, K. New approach relevant to total heat transfer coefficient including the effect of radiation and convection at the ceiling in a cooled ceiling room. Applied Thermal Engineering, 29 (8-9), 1561-1565.

[5] Jeong, J.W. Mumma, S.A. Bahnfleth, W.P. Energy Conservation Benefits of a Dedicated Outdoor Air System with Parallel Sensible Cooling by Ceiling Radiant Panels, ASHRAE Transactions; Atlanta Vol. 109, (2003): 627-636.

[6] Chiang, W.H. Wang, C.Y. Huang, J.S. Evaluation of cooling ceiling and mechanical ventilation systems on thermal comfort using CFD study in an office for subtropical region, Building and Environment, Volume 48, February 2012, Pages 113-127.

[7] Xie, D. Wang, Y. Hanqing, W. Shunquan, M. Maili, L. Numerical analysis of temperature non-uniformity and cooling capacity for capillary ceiling radiant cooling panel, Renewable Energy, Volume 87, Part 3, March 2016, Pages 1154-1161.

[8] Yin, Y. L. Wang, R. Z. Zhai, X. Q. Ishugah, T. F. Experimental investigation on the heat transfer performance and water condensation phenomenon of radiant cooling panels, Building and Environment, Volume 71, January 2014, Pages 15-23.

[9] ASHRAE, HVAC Systems and Equipment (SI), (2008).

[10] Causone, F. Corgnati, S.P. Filippi, M. Olesen, B.W. Experimental evaluation of heat transfer coefficients between radiant ceiling and room, Energy and Buildings, 41 (2009)

[11] Su, L. Li, N. Zhang, X. Sun, Y. Qian, J. Heat transfer and cooling characteristics of concrete ceiling radiant cooling panel, Applied Thermal Engineering, ATE, 84 (2015)

[12] Karathanassis, I.K. Papanicolaou, E. Belessiotis, V. Bergeles, G.C. Flow and Temperature Fields in Cooling Devices with Embedded Serpentine Tubes, Numerical Heat Transfer, Part A: Applications 66 (4), 349-378, 2014.

[13] Menter, F.R. Two-equation eddy-viscosity turbulence models for engineering applications, AIAA J, 32, 1598–1605 (1994).

[14] Sikula, O. Hirs, J. Analysis of the dynamic behavior of the cooling ceiling in an office room with a view of thermal comfort, ventilation and comfort strategies, 3294- 3302, Proceedings The 23rd IIR International Congress of Refrigeration, 1-8, (2011) .

[15] Fredriksson , J.Sandberg , M. Moshfegh.B. Experimental investigation of the velocity field and airflow pattern generated by cooling ceiling beams. Building and Environment Volume 36, Issue 7, Pages 891-899 (2001).

[16] Catalina ,T. Virgone,J. Kuznik,J. Evaluation of thermal comfort using combined CFD and experimentation study in a test room equipped with a cooling ceiling, Building and Environment

Volume 44, Issue 8, Pages 1740-1750 (2009).

[17] Ansys Fluent 13.0.0, Ansys Fluent Inc., (2010)

[18] Patankar, S. V. Numerical Heat Transfer and Fluid Flow, McGraw-Hill, New York, 1980.