Numerical simulation of radiant ceiling panels for indoor cooling

Giuliano Cammarata, Giuseppe Petrone*, Filippo Masi
University of Catania, Department of Industrial Engineering, Viale A. Doria, 6 – 95126 Catania, ITALY

*Corresponding author: gpetrone@dii.unict.it

Abstract. The aim of this work is to analyse the thermal performance in cooling of two different models of radiative ceiling panels, by varying the discharge temperature and the velocity of heat-carrying fluid. The first one geometrical configuration refers to a coil-embedded panel, while the second one is made by a matrix of tubes. Numerical simulations were carried-out by using a FE-approach to solve governing equations for the physical system. Thermal performances were analysed as a function of the inlet temperature and the inlet velocity of the fluid. Then interpolation functions are proposed in order to assess thermal performances for both geometrical configurations against several working conditions.

1. Introduction

The idea of using the pavement as an endpoint for giving way to heat dates back to more than two thousand years. The Chinese, Egyptians and Romans achieved heating systems inspired by such an idea. In the last decades the heating and cooling radiative panel systems are spreading. In these last years the heating and cooling radiative panel systems are spreading. Practically the usual indoor terminal elements are replaced with panels built by means of pipes appropriately inserted into floors, walls or ceiling installed in order to shape a radiative surface. The ceiling panels are able to provide higher performances than those of the floor during both heating and cooling. The trend of reduction of the water temperature in the heating systems has proceeded to go to input temperatures in heating around 35°C; such a result has been reached both from the point of view of the generation and from the point of view of the elements of exchange coupled with these technologies.

Several studies were carried-out on this topic. Among those, Su et al. [1] studied heat transfer and cooling characteristics of the concrete ceiling radiant cooling panel. Two-dimensional mathematical model of steady state heat transfer was developed by using finite difference method. Heat transfer in the concrete panel was numerically simulated. Miriel et al. [2] experimentally and numerically investigated performances, thermal comfort and energy consumptions of radiant ceiling panel heating–cooling systems. Diaz et al. [3] exploited some results of an experimental study to develop a computational model of cooling ceiling systems. Tye-Gingras et al. [4] developed a CFD model of a room coupled with a semi-analytical radiant panel model to study a heating ceiling and wall hydronic radiant panel system in a typical residential building. They also performed a multi-objective optimization investigation for energy consumption and comfort. Causone et al. [5] investigated in order to assess the heat transfer coefficients between radiant ceiling and room in typical conditions of occupancy of an office or residential building. Internal gains were therefore simulated using heated cylinders and heat losses using cooled surfaces. Evaluations were developed by means of experimental tests in an
environmental chamber. A simplified calculation method for estimating heat flux from ceiling radiant panels was proposed by Okamoto et al. [6] using pipe density on panels and the temperature difference between the room air and the supply water. Their proposed method was validated by experimental data. Using of different energy source as feeding for ceiling radiant panels was recently studied by Bojić et al. [7] that highlighted the opportunity of energy saving and environmental benefit related to that opportunity. Li et al [8] shared some experimental detected results, providing useful heat transfer coefficients for this kind of application. In this framework of studies, the present work is devoted to numerically analyse the thermal performance in cooling of two different models of radiative ceiling panels. Models were built-up exploiting a FE-based commercial software and results were carried-out for several working conditions considered for both geometrical configurations. As final product of our investigation, some interpolation functions are proposed in order to assess thermal fluxes with respect to different functional conditions concerning the working fluid.

2. Modelling

The geometry of the studied systems are presented in Fig. 1a-b. Panels are made of multiple layers of which the back one, in both cases, is made of expanded sintered polystyrene (EPS), a material assuring good thermal insulating properties ($\lambda=0.034$ W/(m K)). In general, its use finds justification because of the opportunity of avoiding the space above the suspended ceiling also being cooled, limiting the final efficiency in that way. The back-side insulating panel turns out to be, in both cases, carefully shaped as to host the active element of the thermal exchange with a different geometry for the two models. In the first case (Fig. 1a) it consists in a coil (“coil panel”), in the second one (Fig. 1b) it is made by an array of radiating tubes joining the inlet to the output collector (“array panel”).

![Figure 1. Geometry of numerical models: “coil panel” (a) and “array panel” (b).](image)

Another significant constitutive element is the aluminum diffuser, that is present in both analysed geometries, even that different thickness and arrangements were considered. The high thermal conductivity of this element allows a remarkable increasing in panels performance. The last layer constituting the panels is a “finishing” layer guarantying closing and protection with respect to the external environment. It also assure the overall aesthetics of the panels. In our geometrical models we choose prefabricated plasterboard for the first panel and a plaster of lime and gypsum in the second case. The details concerning dimensions and shape of the two investigated models of panel, as well as the
arrangement of the different layers that make it up, are graphically reported in Fig. 2, where cross-sections of the two panel configurations are shown.

In order to solve fluid-dynamical and thermal distribution on the 3D numerical models built-up for simulating the cooling panels, the Navier-Stokes and energy equations were numerically integrated in Comsol Multphysics environment, a FE-based software allowing to solve coupled PDE’s equations [9].

![Figure 2. Longitudinal cross-section (portion) of “coil panel” (a) and “array panel” (b).](image)

Under assumptions of Newtonian fluid and incompressible flows, Navier-Stokes and energy equations were discretized on no-structured and no-uniform numerical grids made of tetrahedral second order elements. A standard turbulence k-ε method [10, 11] was applied to solve the flow by an eddy-viscosity approach. Solutions were achieved by applying an iterative procedure based on a modified Newton-Raphson method [12]. At each computational step, a direct solver is applied to solve linear systems coming from numerical discretization. Governing equations were solved by considering the following boundary conditions: entry velocity (U_IN) and entry temperature (T_IN) for inlet fluid, logarithmic wall function for fluid at solid walls and null pressure at the out-coming sections, thermal insulation on lateral and back side on the panels, thermal flux on the front-face of the panels (facing to the ambient). Considering a conjugate convective and radiating flux to the ambient, a convective heat transfer coefficient (h_c = 13 W/(m^2 K)) and a surface emissivity value (ε = 0.95) were set in the models. Simulations were carried-out for parametric value of U_IN = 0.1-0.5 m/s and T_IN = 16-25 °C. A constant ambient temperature (T_AMB = 26 °C) was considered in simulations. Some test grids were preliminarily carried out in order to adequately refine the computational grid to obtain mesh-independent results.

3. Results

Simulation results are now presented reporting the thermal analysis for both geometries. The performances were analysed according to the variation of the temperature and the velocity of entry of
the working fluid. Fig. 3a-b shows the thermal distribution in a front view of the “coil panel” (faced to the ambient) in two different working conditions. It can be noticed as increasing mass flow rate allows to minimize water temperature difference between inlet and outflow section. This item is due to the minor residence time of the fluid vector in the heat exchanger. Otherwise, this result was attended because of in our simulations we assumed constant heat flux coefficients at the panel/air-room interface. As a consequence, we can deduce that increasing in mass flow rate not only assures increasing in heat flux transferred to the indoor ambient, but also better thermal comfort conditions, related to a much more uniform radiant temperature distribution with respect to the panel surface. In analogy, in Fig. 4a-b thermal maps are reported for the “array panel”. In that configuration, non-homogeneous thermal distribution related to low mass flow rate is less evident. This can be considered a direct consequence of the water distribution system.

![Figure 3](image1.png)

**Figure 3.** Thermal distribution in front-view (ambient-faced) for “coil panel” and $T_{IN}=16 \, ^\circ C$, considering $U_{IN}=0.5 \, m/s$ (a) and $U_{IN}=0.1 \, m/s$ (b).

![Figure 4](image2.png)

**Figure 4.** Thermal distribution in front-view (ambient-faced) for “array panel” and $T_{IN}=16 \, ^\circ C$, considering $U_{IN}=0.5 \, m/s$ (a) and $U_{IN}=0.1 \, m/s$ (b).
Starting from temperature distributions solved by the FE models, values of the specific thermal flux supplied by the panel to the indoor environment were evaluated. A quantitative post-processing analysis of the obtained results was carried-out by determining thermal flux of cooling panels as a function of the inlet velocity and difference between inlet and ambient temperature. Graphics plotted in Figs. 5-6 report computed values for the “coil panel”, while results derived from “array panel” simulations are shown in graphics of Figs. 7-8. From those, some interpolation functions were proposed, allowing to deduce thermal efficiency for both configurations by using as input data the inlet velocity or the inlet temperature of the working fluid.

![Graph](image1.png)

**Figure 5.** Supplied thermal flux to the ambient by the “coil panel” as a function of the inlet velocity of the working fluid.

![Graph](image2.png)

**Figure 6.** Supplied thermal flux to the ambient by the “coil panel” as a function of the difference between ambient and inlet temperature of the working fluid.
Figure 7. Supplied thermal flux to the ambient by the “array panel” as a function of the inlet velocity of the working fluid.

Figure 8. Supplied thermal flux to the ambient by the “array panel” as a function of the inlet velocity of the working fluid.

Interpolation functions used for “coil panel” data set are proposed below, reporting also the obtained regression coefficient (R) for each one:
\[ q = - 106.26 U_{IN}^2 + 94.11 U_{IN} + 46.25 \quad (R=0.984) \]  

(1)

\[ q = 8.0749 \Delta T + 0.0668 \quad (R=1.000) \]  

(2)

In a similar manner, interpolation functions used for “coil panel” data set are proposed below:

\[ q = - 88.23 U_{IN}^2 + 89.34 U_{IN} + 41.97 \quad (R=0.993) \]  

(3)

\[ q = 7.5622 \Delta T + 0.2408 \quad (R=1.000) \]  

(4)

From the proposed correlation laws it is possible to assess the thermal performance of the studied configurations as a function of the feeding fluid inlet conditions.

4. Conclusions

In this paper a numerical analysis is proposed in order to investigate on thermal performance of ceiling panels for indoor cooling in two geometrical configurations. The first one refers to a coil shape of the working fluid tube, the second one refers to an array and collector configuration. From parametric analyses carried-out as a function of the fluid inlet temperature and mass flow rate it was possible to point-out some interesting aspects concerning both configurations. In particular, it was highlighted as the “coil” layout assures good thermal distribution for the higher mass flow rate analysed; otherwise, it appears that this kind of system could determinate non homogeneous thermal distribution if the mass flow rate is low. This item could be cause of potential indoor thermal discomfort. From a quantitative point of view, thermal flux transferred by the “coil” configuration appears slightly higher than this offered by the “array” configuration. As exploitable result of our investigation, we propose a set of interpolation functions in order to assess the influence of the inlet temperature and mass flow rate of the working fluid on the thermal performance of the studied component in both geometrical configurations considered.

References
[1] Su L, Li N, Zhang X, Sun Y, Qian J 2015 Appl. Therm. Eng. 84 170
[2] Miriel J, Serres L, Trombe A 2002 Appl. Therm. Eng. 22 1861
[3] Diaz N F, Lebrun J, André P 2010 Int. J. Refrigeration 33 793
[4] Tye-Gingras M, Grosselin L 2012 Build. and Environment 54 1
[5] Causone F, Corgnati S, Filippi M, Olesen BW 2009 Energy and Build. 41 622
[6] Okamoto S, Kitora H, Yamaguchi H, Oka T 2010 Energy and Build. 42 29
[7] Milorad Bojić M, Dragan Cvetković D, Ljubiša Bojić L 2015 Appl. Energy 138 404
[8] Li R, Yoshidomi T, Ooka R, Olesen B W 2015 Energy and Build. 86 58
[9] Comsol Multiphysics Version 4.3 User’s Guide 2012
[10] Ignat L, Pelletier D, Ilinca F 2000 Comp. Meth. in Appl. Mech. and Eng. 189 1119
[11] Lauder BE and Spalding DB 1974 Comp. Meth. in Appl. Mech. and Eng. 3 269
[12] Deuflhard P 1974 Num. Math. 22 289