COMFORT AND ENERGY CONSUMPTION OF HYDRONIC HEATING RADIANT CEILINGS AND WALLS BASED ON CFD ANALYSIS

Maxime Tye-Gingras, Louis Gosselin[1]

Département de génie mécanique, Université Laval, Québec City (QC), Canada, G1V 0A6

Article accepté pour publication dans : Building and Environment, Volume 54, August 2012.

Abstract

This article presents the methodology and results of a hybrid numerical optimization study of a heating ceiling and wall hydronic radiant panel system in a typical residential building located in Quebec City, Canada. The comfort and energy consumption of the system are the two figures of merit that are considered in the multiobjective optimization analysis. The main design variables are the position and dimension of the panels, and the fluid inlet temperature. The hybrid numerical method features a 2D CFD model of a typical empty room, coupled with a semi-analytic radiant panel model specially developed for coupling with CFD. This strategy allows considering the real room geometry, while providing at the same time accurate temperature profiles of the radiant panels and detailed temperature and comfort data field in the room. The results show that there is no unique optimal solution but rather a family of optimal designs (Pareto fronts) for which the solutions are trade-offs between the two objectives. When adjusting correctly the fluid inlet temperature, it is also possible to achieve nearly Pareto optimal solutions, even when reducing the total panel surface by 66%. This means that the

[1] Corresponding Author: Louis.Gosselin@gmc.ulaval.ca; Tel. 418-656-7829; Fax 418-656-7415.

temperature control of the fluid is the most important parameter for maximizing comfort and minimizing energy consumption of hydronic heating radiant panels.

Keywords: radiant hydronic panel; thermal comfort; energy consumption; CFD; modeling; Pareto optimization

Nomenclature

A	area, m^2
C	heat capacity rate, W/K
D	diameter, m
F	coefficient for heat transfer equation
GDQ	global dissatisfaction quote
H	height, m
L	length, m
Nf	number of faces
PPD	predicted percentage of dissatisfied
PMV	predicted mean vote
Q'	total heat transfer rate per unit depth of the room, W/m
R'	thermal resistance per unit length, mK/W
R''	thermal resistance per unit surface m^2K/W
SARPM	semi-analytic radiant panel model
T	temperature, K
U	overall heat transfer coefficient, $W/m^2\,K$
W	center-to-center tube spacing, m
k	thermal conductivity, $W/m\,K$
h	convection coefficient, $W/m^2\,K$
m	fin parameter, m^{-1}
$\dot{m}'$	mass flow rate per unit length, $kg/s \cdot m$
n	tube row index
q	heat transfer rate, W
t	panel thickness, m
$\bar{v}$	air velocity magnitude, m/s
x,y,z	Cartesian coordinates of the room, m
y', z'	Cartesian coordinates of the radiant panel, m

Greek Symbols

Γ	non-dimensional temperature
ε	surface emissivity

η efficiency

θ temperature difference $T - T_\infty$, K

χ, ξ local coordinates, m

ω $(W - D_o)/2$, m

Subscripts

C cold
H hot
c ceiling
f fluid
fin fin
i tube inside
in inlet
n tube row index
o tube outside
opt optimal
out outside
p panel
r room
rad radiant
ref reference
t tube
tot total
w wall
win window

Superscripts

f cell face

1. Introduction

Hydronic radiant heating and cooling panels are now commonly used in new commercial buildings and their popularity is still increasing [1]. As a consequence, there has been a growing interest in the last years for characterizing the heat transfer performances and the thermal comfort provided by such systems. Over the years, several different approaches have been used by authors to carry out this task.

In Ref. [2], the thermal comfort and energy consumption of a room equipped with radiant cooling and heating ceilings is studied experimentally. Hourly annual simulations using a simplified model on the energy simulation software TRNSYS were performed for characterizing the total energy consumption of the system during a year. Another large scale simulation was undertaken in [3] for optimizing building parameters, including heat exchange area of radiant panels. The objective was to minimize the energy consumption and maximize the comfort in an entire building. The authors realized building energy simulations with *EnergyPlus*, that was sequentially called by a genetic algorithm for parameter optimization.

A field study analysed the occupants comfort vote in a typical large building equipped with radiant floors [4]. Results showed that the comfort results obtained for a real population is in accordance with the PMV model [5]. Participants generally reported reduced thermal discomfort with radiant surfaces system. The thermal comfort and energy consumption of heating and cooling radiant panels were also investigated and compared with that of an all-air system in another experimental campaign [6]. The comfort was evaluated by the votes of human subjects, and the radiant panels proved to yield a more comfortable environment than the all-air system. A numerical model of typical rooms was used for computing energy consumption in different cases, but the model was not developed in the paper. The authors reported up to 10% energy savings with radiant systems compared to conventional ones.

Thermal comfort and energy consumption of a high temperature radiant stove in an occupied room is numerically analysed in [7]. The authors used a 3D finite element model for computing view factors, and the thermal balance and comfort of the occupant was computed analytically according to the PMV-PPD model. The results showed a strong influence of the stove position on the comfort and the energy consumption.

An experimental and numerical CFD study of thermal comfort of a radiant cooling panel system is presented in [8]. The authors used CFD for obtaining the air velocity fields in a room, but the radiant temperature field was computed separately from CFD with the radiosity method. The temperature field agreement between CFD and experiments was satisfactory. Ref. [9] also used experimentally validated CFD simulations for comparing the comfort (Predicted Percentage of Dissatisfied people, PPD) and energy consumption of different heating systems, including radiant floor and wall, for an office room in Swedish climate. The radiation was handled by a surface-to-surface method, but no explanation was provided regarding how the mean radiant temperature field was obtained for computing the PPD field. The energy consumption and the PPD field in the room were computed for each heating system, but no global metric was established for conveniently comparing their performance.

Some conclusions can be drawn from the review of the above mentioned studies. First, as noted by Diaz et al.[10], it is necessary that the performance of cooling hydronic ceilings be evaluated while coupled to the building system and structure. Thus, the thermal comfort and heat transfer computation for these systems cannot be accomplished using standard zone heat balance models, for which the accuracy depends on averaged air temperature and heat transfer coefficients that are based on idealized and highly controlled conditions (e.g. uniform temperature profiles). Instead, the heat transfer and comfort analysis should rather be made using a comprehensive model, incorporating every aspect of the problem, i.e. natural convection patterns in the room, wall boundary layers, radiation field, temperature field, etc.

Second, CFD is now seen as a reliable tool for computing air velocity and temperature fields in rooms. Thus, there is an interesting potential for using CFD to simulate radiant panels systems. However, in order to really take advantage of CFD, the models should be comprehensive, (i.e. include radiation and incorporate the local temperature or heat flux of the panels). However, most of the radiant panel models currently described in literature [11-18] consider averaged properties over the panels, and are thus not optimal for coupling with CFD, that provides detailed air flow, temperature, and radiation intensity fields. The radiant panel model proposed in this study solves that problem by considering local properties and yielding a detailed temperature field. Finally, one of the major advantages of a fully numerical model over an experimental apparatus is to be modifiable easily and at virtually no cost. This means that a rigorous radiant system optimization is possible with CFD.

Third, the majority of the studies undertaken considered the radiant panels in heating and cooling, a feature that could be exploited in regions experiencing cold winters and hot summers (e.g., many areas of Canada). However, in some regions, the heating load is often more important than the cooling load, especially in residential buildings. In that case, the design of the systems should be based primarily on the heating mode, i.e. assuming outside cold winter conditions. Most of the radiant systems currently installed in this type of climate are auxiliary heating systems that are not designed to handle the full heating load of a typical residential building. In order for the hydronic radiant panels to act as primary heating systems, they must be able to provide enough heat to the buildings, minimize heat losses to the exterior and provide a high level of comfort to the occupants. There is thus a need for optimizing the geometry of the panels, their position in the building, and their operating parameters.

Considering the above-mentioned observations, this article presents a numerical study of a room of a typical residential house heated by hydronic radiant panels in the winter design conditions of Québec City, QC, Canada. The methodology employs a semi-analytical radiant panel model (SARPM) [19] that is directly coupled with CFD calculations in the room to simulate the real interaction between each panel and its environment. The purpose of the article is to show the impact of the size, position, and fluid inlet temperature of the panels on thermal comfort and energy consumption of the system. The problem will be assessed in the form of a multiobjective optimization procedure to minimize heat losses through the envelope and predicted percentage of dissatisfied people (PPD), i.e. maximize comfort.

2. Description of the system

Figure 1 presents the system considered. It is a room with one façade wall (left of Fig. 1), of a residential building located in Québec City, QC, Canada. The room is heated by hydronic radiant panels that can be embedded in the ceiling and/or the right wall. Note that y' and z' refer to the panel coordinates while x,y,z refer to the room coordinates. The z'- axis is always parallel to z, while the axis y' can be parallel to x or y:

$$\left(y',z'\right) \mathrm{A}\left(x,z\right) \text{ for ceiling panels} \tag{1a}$$

$$\left(y',z'\right) \mathrm{A}\left(y,z\right) \text{ for wall panels} \tag{1b}$$

The room is modeled in two dimensions (x,y), although the radiant panels tubes stretch along the z-dimension. This assumption is possible because the temperature gradients on the panel along the z'-dimensions are low compared to the ones in the y'-dimension [19]. The number of panels, their length L_p and their position shown in the figure is arbitrary, as these will vary throughout the study. Only their height H_p and their orientation are uniform and constant.

The winter design conditions are applied outside [20]. As in standard heating load calculations, solar irradiation and internal gains are assumed to be null. Room characteristics climatic data and material thermal specifications applying throughout the study are summarized in Table 1. A further description of the model assumptions for the room and the panels system is provided in §3.2 and 3.3.

Serpentine-shaped hydronic radiant panels are used. In previous studies, the serpentine tubing has shown heat transfer rates similar to parallel flow tubing panels, while being technically simpler [19][21]. This is why this tubing configuration is chosen for the present work. The size, the position, and the fluid inlet temperature of the panels are the design variables, while all the other panel characteristics are identical for every panel and are given in §3.3.

3. Modeling of the system

3.1 The coupled iterative procedure

The major feature of the proposed model is the coupling between CFD in the room and the radiant panels. Fig 2a schematizes the interaction between the two modeling stages and Fig. 2b describes the iterative procedure used to achieve the convergence of calculations. The case shown is for a single ceiling panel but, as mentioned earlier, there can be multiple panels and they can be embedded in the wall as well as in the ceiling.

First, the temperature of the radiant panels is initialized. This provides boundary conditions for solving the conservation equations using CFD, which allows the calculation of a local heat transfer coefficient at each wall cell. These results are imported into a semi-analytical hydronic radiant panel model (SARPM) to compute the new temperature distribution on the panel at each point corresponding to a cell of the CFD mesh. This provides new boundary conditions for CFD computations. The sequence is repeated until convergence is reached. A more detailed description of the modeling methods and validations is provided next in §3.2 and §3.3.

There are three convergence criteria applied to the iterative procedure. The first and second consider the stability of the two objectives. First, the comfort metric, expressed in a global dissatisfaction quote (GDQ) is allowed an absolute variation of 0.1% between two consecutive iterations. The calculation method of the GDQ will be explained in §3.5. Second, the variation of the total heat transfer rate per unit depth of the room through the envelope Q'_{out} ,must be under 0.5 W/m between two consecutive iterations. The third convergence criterion verifies the adequacy between the heat transfer rate given by the SARPM and the CFD. A total maximal discrepancy of 0.5 W/m is tolerated.

Once the solution is converged, the surface heat transfer rate on the façade wall and window gives the total heat losses per unit depth of the room through the envelope Q'_{out} . Also, knowing the air velocity, temperature and radiation field intensity distribution, the thermal comfort in the room can be computed. The comfort model used is described in §3.4. It is worth to recall that the two output metrics of the study are the thermal comfort and the heat dissipation rate (energy consumption).

<u>3.2 CFD room modeling</u>

The commercial finite volume code ANSYS FLUENT is used for the 2D CFD calculations. Steady-state mass, momentum and energy conservation equations are solved. For the room size considered and the typical temperature difference expected, the Rayleigh number based on the height of the room can reach $Ra \approx 10^{10}$, which means that turbulence is to be expected. The turbulence closure model chosen is the shear stress transport (SST) k-ω model [22][23]. This model features a blending function that solves the wall-bounded flows using the standard k-ω model in the boundary layer and the k-ε model in the far field. This strategy was chosen because the flow is not expected to be fully turbulent near the walls, in which case the k-ω is proven to be efficient. Note that both the standard k-ω and k-ε models were tried, yielding harder convergence and worse agreement with literature results that will be presented in the next section (validation).Note that a fully laminar model was also tested, yielding considerably different flow patterns compared to literature [24], justifying once again the use of a turbulent model.

Since it is desired to obtain the radiation field as an output, the equation of radiative heat transfer is solved in the whole room, treating air as a participating media with an extinction coefficient $\beta = 0$. The radiation model chosen is the discrete ordinates method (DO) [25] because it is recognized for its accuracy in optically thin media [26]. The angular discretization is set to 4 directions per octant.

The effect of ventilation on the flow pattern and on the heat load is disregarded. In other words, only the airflow triggered by buoyancy forces is considered. This choice is made to keep the results as general as possible and considering the fact that most of the residential buildings in Quebec City, Canada do not have a ventilation system, i.e. they rely on infiltrations to provide makeup air.

As mentioned previously, the simulation is performed for steady-state winter design conditions, which consists in the worst possible case. The walls in the CFD model are thus assumed to have negligible thermal mass. The interior wall, floor and ceiling are adiabatic and the façade wall is given a medium insulation value of $R''_w = 3\,m^2K/W$. The window, occupying the superior half of the façade wall, is double glazed with a 6mm argon space and has a low-e coating on one of the interior glass faces. The overall heat transfer rate through the window (excluding outside house and inside room radiation and convection) is set to $U_{win} = 3.07\,\mathrm{W/m^2K}$. This value was chosen so the heat flux through the window corresponds to that of a similar system in which the total U value including the outside and inside radiation and convection resistances is $U_{win} = 2.07\,\mathrm{W/m^2K}$ as given in Ref. [20]. The exterior heat transfer coefficient, considering forced convection due to wind and radiation due to the atmosphere, is $34\,\mathrm{W/m^2K}$ [20] for both the wall and window. The interior heat transfer coefficient is not imposed because the wall-to-indoor space heat transfer is modeled by CFD. All the interior surfaces of the room (including interior side of façade wall, window, and radiant panels) are given a value of emissivity $\varepsilon = 0.9$, which is typical in building heat transfer applications.

Finally, the radiant panels can be embedded into the ceiling and/or the right wall, occupying either the entire surface or only a part of it. In the CFD code, a temperature boundary condition is imposed on each cell face of the boundary zones corresponding to the radiant panel. The temperatures are directly imported from a panel discrete temperature field provided by the semi-analytic radiant panel model (see §3.3). The details of the temperature coupling procedure are also given in §3.3. The portions of the ceiling and right wall that are not covered by radiant panels are adiabatic.

The outputs of the CFD calculations are used in two different applications. First, the local surface heat transfer coefficients at the boundary of the panels must be exported into the semi-analytical radiant panel model. For each boundary cell face of the CFD mesh, the local panel surface heat transfer coefficient, incorporating convection and radiation, is computed as follows:

$$h_p^f = \frac{q_p^f}{A_p^f\,\theta_p^f} \tag{2}$$

where q_p^f is the total heat transfer rate through the cell face boundary of the panel and $\theta_p^f = T_p^f - T_{ref}$ is the temperature difference on which is based the heat transfer coefficient. The reference temperature T_{ref} is fixed at 293K, but this value is arbitrary and has no effect on the final result, since the same reference temperature and heat transfer coefficients are used for heat transfer computations in the radiant panel model (see §3.3). The coefficients h_p^f are updated at each iteration of the procedure described in §3.1. Knowing the position of the panel cell face centroïds $(x, y)_p^f$, the coefficients h_p^f are directly converted into a discrete profile $h(x, y)$ on the walls. It can then be exported into the SARPM.

The second type of outputs is used for computing the heating load and thermal comfort at the end of the iterative process, when convergence is reached. The CFD code provides the air velocity magnitude $\overline{v}(x, y)$, the temperature $T(x, y)$ and the radiation temperature $T_{rad}(x, y)$ distributions inside the room as well the total surface heat transfer rate to the outside Q_{out} (i.e. through façade wall and window).

An unstructured triangular mesh is chosen in the central zone of the domain, because it can adapt easily to the geometry changes caused by displacing panels on the boundaries. The total number of cells is around 3000 for all the simulations. The grid independence was tested for the system described in §2, but with an imposed constant temperature of 300K on the ceiling and the right wall. The mesh was refined by halving the size of the cells until the total heat transfer rate of the domain changed by less than 0.1%. The mesh near walls and panels is rectangular, with an exponential spacing increment that was adjusted to provide sufficient precision in the viscous sub-layer of the turbulent boundary layer, with at least 5 cells within $y^+ < 11.5$ [24].

Two criteria are set for judging convergence of each CFD simulation. First, the scaled residuals must be under 10^{-3} for all the governing equations, except energy and DO intensity equations of which the residuals must be under 10^{-6}. In addition, the total heat transfer rate imbalance in the model must be under 0.5% of the total heat transfer rate leaving the domain.

3.3 Validation of the CFD model

The model used to solve the fundamental equations was chosen for its ability to produce the desired outputs, its stability and convergence behavior, and its concordance with both experimental and numerical data found in literature. The code was validated using the numerical data of Xu et al. [24] and the experimental data of Olson et al. [27]. Both papers show temperature results for a 2D room of dimensions $H_r = 2.5m \times L_r = 7.9m$ with differential wall heating (cold wall with temperature T_C on the left and hot wall with temperature T_H on the right). All other surfaces are adiabatic. The geometry and the Rayleigh numbers are similar to that of the present study. Figure 3 shows the comparison with numerical (LB model from Ref. [24]) and experimental data. The variable Γ is the non-dimensional temperature $\Gamma = (T - T_C)/(T_H - T_C)$. The plots indicate good agreement of the temperature profile near the walls and in center zone of the room.

The precision of the radiation DO model with 4 directions per octant was verified by comparing the results for the same case with a surface-to-surface model that disregards the presence of air in the room, as it is commonly done in radiation calculations in buildings. The results yielded less than 0.01% difference, which confirms the ability of the DO model to represent radiative heat transfer in the zone. We recall that such a model is necessary to obtain the radiation temperature field everywhere in the room, for comfort calculations.

3.4 Radiant panel modeling

Figure 4 shows the serpentine-shaped hydronic panel used in this study. A longitudinal coordinate system (χ, ξ) is used for calculations, with χ following the tubing and ξ giving the inter-tube position (always in the positive y' direction). The coordinates χ and ξ are related to z' and y' with the variable n, the tube row index.

As stated earlier, the panels can be embeded in the ceiling and/or the right wall. They are oriented in such way that the tubes are parallel to the z-axis of the room, as shown in Fig. 1. The fluid inlet location on each panel is set at $y' = 0$ by default but the effect of permutation (i.e. inlet located at $y' = L_p$ instead of $y' = 0$) will be analyzed.

The total water mass flow rate per unit depth of the room is given a default value of $\dot{m}' = 0.05\,\mathrm{kg/m \cdot s}$ and it is split between all the panels of the system proportionally to their length L. This methodology allows excluding pumping power considerations from the problem. It also allows extending the results to any panel length in the z'-direction, because the temperature drop in the y'-direction is dominant and due to the fixed mass flow rate per unit of depth, the temperature profile becomes almost independent on L_p (see Eq. (10) in Ref. [19]). The effect of modifying the mass flow rate on the heat transfer and thermal comfort results will also be briefly discussed further.

The position, the length L_p, and the fluid inlet temperature are the main parameters that are subject to variations throughout the present study. The other parameters of the panels are constant and are shown in Table 2.

The hydronic radiant heating panel model used is the semi-analytical model described in Ref. [19] with a few adaptations to suit the type of calculations performed in the present study. The model includes the major following assumptions:

- No thermal gradient through the thickness of the pannel
- No thermal mass in the pannel
- No backloss (panel perfectly insulated)
- 1D (y'- direction only) heat conduction in the panel
- Thermal symetry between tubes
- Semi-circular tube endings neglected

The three last assumptions of the above list were validated in [19]. This allows using analytical solutions developed for fins to calculate the local heat transfer on every fin-like element of the panel. The local heat transfer rate is related to the temperature drop, which yields the following differential equation:

$$-C\frac{d\theta_f(\chi)}{d\chi}=\frac{\theta_f(\chi)}{R'_{tot}\left(1+1/F(\chi)\right)} \tag{3a}$$

with

$$F(\chi)\equiv R'_{tot}\left[(\eta_{fin}\omega)\left(U_{p,left}(\chi)+U_{p,right}(\chi)\right)+D_o U_{p,t}(\chi)\right] \tag{1b}$$

$$\eta_{fin}=\frac{\tanh(m\omega)}{m\omega} \quad \text{and} \quad m(\chi)\equiv\sqrt{U(\chi)/k_p t} \tag{1c}$$

where C is the heat capacity rate, $\theta_f = T_f - T_{ref}$ is the relative temperature of the fluid, R'_{tot} is the total thermal resistance between the fluid and the fin base (where the tube touches the panel), and η_{fin} is the fin efficiency of each panel slice, considering adiabatic fin tip located at the symmetry axis between each tube row.

The variables $U_{p,left}(\chi)$, $U_{p,right}(\chi)$ and $U_{p,t}(\chi)$ are the overall heat transfer coefficients of the panel for the left fin, right fin, and under-tube section of each tube row. In order to keep the semi-analytical methodology based on the fin solution, each fin slice must have a uniform U_p coefficient. However, this coefficient is allowed to vary according to the coordinate χ following the tubing. In this study, the local U_p coefficients are calculated based on the h_p^f values provided by CFD (see Eq.(1)). The treatment applied to convert the heat transfer rate data from the CFD to the radiant panels will be detailed later.

Since the coefficient F is a discrete unknown function of χ, there is no analytical solution to Eq.(1). A forward finite difference scheme is applied for solving $\theta_f(\chi)$ along the tubing, starting from the known inlet fluid temperature $T_{f,in}$. The temperature at each location (y', z') of the panel is then retrieved by using analytical expressions for temperature distribution in each fin.

The U data exported from the CFD must be converted so that it can be used in the SARPM. The first conversion is required due to the introduction of the 3rd dimension by passing from the 2D CFD plane (x, y) to the 2D SARPM plane (y', z') (see Fig. 1). This is handled by assuming that U coefficients are uniform in the z' (or z) - direction. The rest of the resolution is done normally, i.e. considering the 3rd dimension.

The second conversion treatment that is required consists in averaging the $h_p^f\ Ah(y')$ coefficients provided by CFD over each fin element to create the $U_{p,left}(\chi)$, $U_{p,right}(\chi)$ and $U_{p,t}(\chi)$ coefficients necessary for solving Eq. (1). Because of the loss of the 3rd dimension discussed above, these are in fact step-type functions, being constant for each tube row. The h_p^f coefficients collected at the CFD boundary cell faces corresponding to each section (left fin, right fin and under-tube) of each tube row of the panel are processed to yield a weighted average that is calculated as follows:

$$\overline{U_p} = \frac{\sum_{f=1}^{Nf} \theta^f A^f h_p^f}{\sum_{f=1}^{Nf} \theta^f A^f} \tag{4}$$

where the superscript f is the cell face of the CFD mesh and Nf is the total number of cell faces located in each averaging section of the panel in the SARPM.

Finally, after the SARPM is resolved, a modeling plane switch between the two simulation levels needs to be handled again, because the SARPM yields a temperature field in the plane (x,z) or (y,z), contrary to (x,y) for CFD. Since it was concluded in [19] that the temperature gradient of the panel according to z' is generally not significant (especially at medium and high flow rates), it is possible to convert the 2D panel temperature field into a 1D temperature profile, by taking the temperature values at the middle height of the panel:

$$T(y',z') \rightarrow T(y')\big|_{z'=H_p/2} \tag{5}$$

The temperature coupling is then made with a user-defined function in the CFD code (UDF) that finds the closest available temperature value at each boundary cell face center. The boundary cell face temperatures are then simply pluged as boundary conditions into the CFD model to be solved.

3.5 Thermal comfort calculations

Comfort is evaluated according to the method developed by Fanger [5]. The model considers a number of parameters related to the environment and the occupants. Only the air velocity, the temperature, and the radiant temperature are taken as variables in this study and are provided by the CFD outputs. The other parameters and their associated values are presented in Table 3.

The PMV (predicted mean vote) and PPD (predicted percentage of dissatisfied) can be computed at any location of the room to obtain the comfort field. This PPD field can be space-averaged to yield a global comfort index in the room.

In addition to the traditional PPD as a measure of thermal comfort, Fanger introduced the LPPD (lowest PPD) as a measure of the thermal non-uniformity in a zone. It is defined as the lowest PPD that could be reached by making an overall offset of the thermal comfort vote to reach $\overline{\mathrm{PMV}} = 0$. That is:

$$\mathrm{LPPD} = \mathrm{PPD}\left(\mathrm{PMV} - \overline{\mathrm{PMV}}\right) \tag{6}$$

For example, in a room with uniform conditions, $\overline{\mathrm{LPPD}}$ will be at its minimum value (5%), but $\overline{\mathrm{PPD}}$ might be large if the predicted general sensation of the occupants is too cool or too hot. On the other hand, the $\overline{\mathrm{LPPD}}$ will never be higher than the $\overline{\mathrm{PPD}}$ but the two metrics can be identical if the conditions are such that $\overline{\mathrm{PMV}} = 0$ (i.e. if the space-averaged thermal comfort is optimal).

Finally, since the minimum value for PPD and LPPD is 5% (there will always be at least 5% of dissatisfied people, no matter the conditions), Fanger recommends the use of $\overline{\mathrm{PPD}} - 5\%$ and $\overline{\mathrm{LPPD}} - 5\%$ to assess the thermal comfort and the thermal uniformity respectively. In this study, those two objectives are grouped into a Global Discomfort Quote (GDQ) that is defined as:

$$\mathrm{GDQ} = \overline{\mathrm{PPD}} + \overline{\mathrm{LPPD}} - 10 \quad [\%] \tag{7}$$

In order to avoid undesired outlier comfort data of unoccupied areas of the room the areas comprised within 5 cm from the side walls and 25 cm from the ceiling are not taken into account in the space-averaged value of the comfort metrics $\overline{PPD}$ and $\overline{LPPD}$.

4. Thermal comfort and energy consumption results and discussion

4.1 Typical results for a specific case

This section presents the main output results of the simulations realized for a specific combination of the main variables (panel length, position and fluid temperature). These results show the type of rough data that will be interpreted and analysed in a more comparative fashion in the next sections. The results shown here are for a system composed of a single panel of length $L_p = 2m$ located in the middle of the ceiling ($x_{in} = 1m$) and with a fluid inlet temperature of 310.5K. The temperature was chosen so that the GDQ is minimal (maximal comfort).

Figure 5 presents respectively a) the room temperature field, b) the room radiant temperature field and c) the room PPD and PMV field, after convergence, computed by the CFD software. In c), the PPD is shown with the contour lines and the PMV is given by the color scale, which allows to visualize the relation between them. For example, we can see that the PPD (vote for thermal dissatisfaction) increases either if the PMV (the predicted sensation of cold or hot) gets too high or too low, the neutral value 0 being the best. Again in c), the dashed line rectangle shows the zone within which the space-averaged values of the comfort metrics are calculated. Finally, Figure 5d gives the converged panel temperature profile, computed by the SARPM, which was used as a boundary condition for CFD calculations. In d), the dashed line indicates the axis on which the temperature values are taken for exportation to the CFD model.

In Fig. 5a, a temperature stratification caused by the ceiling heating panel is clearly observed, but the temperature gradient between head ($y = 1.1m$) and ankle ($y = 0.4m$) level is less than 1K, which is under the standard limitation of 3K set by

ISO7730:1994 [28]. The thermal boundary layer close to the window is especially thin compared with that on the interior vertical wall, which gives a slight non-uniformity of the temperature field. A small hot spot is also observed at the bottom right corner, but it is only the results of a recirculation zone where the air is warmed up by the floor and wall. Nevertheless, the horizontal uniformity of the temperature profile is globally good, considering that the panel occupies only half of the room length.

On the other hand, the radiation temperature profile in Fig. 5b is less uniform and shows larger amplitude than the air temperature field. The low radiant temperature spot observed close to the window occupies a considerable area, while there was no such low temperature spot in Fig. 5a. This happens because the radiant intensity field is not influenced by the air flow that drags the cold air down the façade wall, but by the cold temperature of the window itself. The zones located at proximity of the interior surfaces (floor and right wall) generally have tempered values of radiant temperature. The zone near the window exhibits the lowest radiant temperature, while the zone near the panel has the largest radiant temperature. This suggests that the more comfortable panel position solution could be found when placing the heating panel near the window, in such a way that the “hot effect” of the panel would counterbalance the “cold effect” of the window. On the other hand, this would create an important radiant directional asymmetry in that zone. However, to preserve the relative simplicity of the model, the radiant asymmetry is not taken into account in the Global Discomfort Quote used as the comfort metric.

The PPD (contour lines) and PMV (color scale) fields in c) reveal a relatively uniform comfort distribution in the room, which will translate into a low value of the $\overline{\mathrm{LPPD}}$. As the radiant temperature field suggests, the PPD increases near the panel and the window. Nevertheless, in the major part of the occupied zone within which the $\overline{\mathrm{PPD}}$ is calculated, the percentage of dissatisfied is nearly minimal, under 5.5%, even with only half of the ceiling surface covered by radiant panels.

The radiant panel temperature field shown in d) demonstrates that the thermal gradients on the panel are very low in the z' direction, which confirms the validity of the 2D simplification in the z direction. The gradients are also considerably low in the y' direction (under 1K over the total length of the panel). However, because the gradients are strongly related to the fluid flow rate, it would be interesting to analyse the results with a different value of the fluid mass flow rate. This will be done in §4.3. Finally, we recall once again the assumption that no sunlight is entering in the room through the window (heating design conditions, i.e. at night). The presence of solar irradiation would influence significantly the results, and especially the radiant temperature field.

4.2 Influence of the fluid inlet temperature, panel length and position

This section discusses the impact of the fluid inlet temperature on thermal comfort and energy consumption of the system for various size and positions of radiant panels. Eight different panel arrangements are tested with a large range of inlet fluid temperature: $295\,\mathrm{K} \le T_{f,in} \le 315\,\mathrm{K}$ by increment of $0.5\mathrm{K}$. The characteristics of the eight panel arrangements are shown schematically in Figure 6. The pictograms show the room in the (x, y) plane with the position of the panel(s) in black. The arrangements are identified by Roman numbers from I to VIII, and the framed number in the center of each pictogram is the total length of the panel(s), that is in fact the total panel surface area per unit depth of the room.

Figure 7a shows the Global Discomfort Quote GDQ as a function of $T_{f,in}$. For the color version of the figure, the reader is referred to the numerical version of the paper. From these results, an optimal temperature set-point for comfort is established for every panel arrangement. The energy consumptions of the eight panel arrangements are then plotted as a function of the temperature gap to their respective optimal temperature $T_{f,in} - T_{f,in,opt}$ in Fig. 7b. Note that the absolute value of Q'_{out} obtained in the simulations considers no infiltrations as well as a fixed set of building parameters and climatic data. Modifying these model assumptions and parameters may affect significantly the flow patterns and heat loads. The emphasis must thus be put on the shape of the curves, more than on the absolute values of Q'_{out}.

The results of Fig 7a show that the influence of the fluid inlet temperature set-point is similar for every panel arrangement, although we observe an important shifting of the curves depending on the panel arrangement chosen. This shifting is partly due to the total panel surface that is not the same in every case, but also to the specific location and orientation (i.e. vertical or horizontal) of the panels that influence the global heat transfer rate. Generally, for a given panel surface area, the wall panels allow using a lower fluid inlet temperature, because their global heat transfer coefficient is higher than the ceiling panels.

The shape of the GDQ curves is such that the temperature must be kept approximately within $\pm 1[\mathrm{K}]$ for the GDQ to remain within 1% (absolute) of its minimal value. The relation between the inlet temperature and the energy consumption is nearly linear, and a maximum relative difference of 6% on the total energy consumption is observed between the various panel arrangements tested.

An interesting conclusion can be drawn by comparing the curves II, III and IV that all show results for a single $L_p = 2[\mathrm{m}]$ ceiling panel, but changing only its position. Although it could seem like an outlier, the curve II simply shows the results of a plausible case demonstrating that the position of the panels can influence significantly the comfort and the energy consumption of the system. Indeed, the decision of installing the panel near the façade wall and window (II), where the heat losses are more important, instead of near the interior wall (IV) allows a relative reduction of nearly 3% of the GDQ but at the same time a 6% relative increase of the energy consumption.

Without surprise, the results show that maximizing comfort and minimizing energy consumption are contradictory objectives, which is typical of most optimization problems involving more than one objective. For example, the minima of Fig. 7a cannot be qualified of “best” inlet temperature since the energy consumption aspects must also be considered. The results must thus be presented and discussed using a Pareto analysis, an approach that is widely used in multi-objective optimization evolutionary algorithms [29-32]. The §5 will present the Pareto analysis of the optimization problem consisting in choosing the best combination for the panels dimensions and positions, as well as fluid inlet temperature.

4.3 Influence of other parameters

The fluid inlet temperature, panels length and position were chosen as the main variables of this study because they were identified as important parameters in former studies [17][21]. However, previous studies did not consider non-uniform heat flux on the panel nor the possibility of flow direction permutation, which might have inhibited the impact of some other parameters. The precision of the model used in this study allows verifying the validity of these conclusions.

In this section, the effect of the permutation of the fluid inlet location and of the mass flow rate is studied. Ref. [21] predicted that a significant modification to the mass flow rate would have only a small effect on the heat transfer rate. However, a reduction of the mass flow rate induces a larger temperature gradient on the panel. If this gradient becomes important, one can expect that the permutation of the fluid inlet location (causing the inversion of the temperature profile) could create a significant change in the thermal comfort field and total heat transfer rate of the system.

Simulations with halved flow rates and/or permutated flow were thus realized for two base cases: a full ceiling (I) and a full wall (VI) panel. The results of the separated and combined effect of modifying these two parameters are shown in Figure 8. The simulation parameters for each of the 8 tests presented are given in Table 4. All the other parameters remain as in Table 2.

Fig. 8 indicates that for both the ceiling (I) and wall (VI) panel arrangements, the influence of halving the flow rate and/or the permutation of the flow direction is negligible, with a maximum relative impact of less than 1.5% on the two computed objectives (GDQ and Q'_{out}). Simulations with other panel arrangements were also conducted, and yielded the same conclusions. Consequently, for the rest of the study, non-permutated fluid flow ($y'_{in} = 0$) and default value of the mass flow rate ($\dot{m}' = 0.05\ \mathrm{kg/s \cdot m}$) will be used.

5. Optimization of the system

5.1 Optimization procedure and Pareto analysis

This section presents the procedure that was applied to optimize the dimension and position of the panels, and the fluid inlet temperature, regarding the two objectives of maximizing comfort and minimizing energy consumption (minimize GDQ and Q'_{out}). Here we considered a random search approach which is quite easy to implement. It proved to work well for this problem because the design space is not too large. Also, it is difficult to describe efficiently the panel arrangements by a small set of independent design variables, which would have been required by other optimization methods. Here is the procedure followed to generate the results:

i) A home-built function randomly generates panel arrangements with an imposed total panel length $L_{p,tot}$. The panels can be embedded only in the ceiling and the right wall, their minimal length is 1m and their maximal length is the one available on surface considered (wall or ceiling). They can take any length in-between by 0.25m increments (1m, 1.25m, 1.5m, etc.). This increment restriction is imposed to simplify the random generation task and to ensure that the number of tubes of the panel is an integer, considering a constant spacing W between the tubes. The total number of panels is chosen randomly but must respect the above-mentioned conditions and the total length imposed.

For example, if we impose a total panel length of 4m, the random generation function could create a panel arrangement featuring 2 ceiling panels and 1 wall panel of respective lengths $L_{p,c1} = 1.25\mathrm{m}$, $L_{p,c2} = 1\mathrm{m}$ and $L_{p,w1} = 1.75\mathrm{m}$ and their position could be $x_{in,c1} = 0.38\mathrm{m}$, $x_{in,c2} = 2.46\mathrm{m}$, $y_{in,w1} = 0.29\mathrm{m}$.

ii) For each randomly generated panel arrangement, a series of fluid inlet temperatures are tested, with increments varying from 1 K to 0.5 K (with the lowest temperature increments around the optimal comfort temperature). The GDQ and Q_{out} results of each design, formed by the combination of a panel arrangement and fluid inlet temperature, are kept in a database for analysis.

iii) When all the data is collected, the designs are sorted in Pareto fronts showing the GDQ and Q_{out} values of the non-dominated designs (i.e. those that are dominated by no one else in both objectives) [33].

Separate data sets are produced for each imposed total panel length $L_{p,tot}$. This allows considering the effect of this variable, that could be related to a cost function, because the price of the system will depend on the surface covered by the panels. However, since it is almost impossible in the case of a theoretical study like this one to find a relevant general cost function depending on the total panel length, it was decided not to include a cost objective, but to compare the results obtained with the different values of $L_{p,tot}$.

Simulations are made for three different values of the total panel length: $L_{p,tot} = [2, 4, 6]\,\mathrm{m}$, with 60 different panel arrangements tested for each case, which makes a total of 180 panel arrangements. With the different fluid inlet temperature tested for each of these, the total number of simulations is approximately 2000. Contrarily to an evolutionary algorithm, the random search undertaken probably doesn't allow reaching fully optimized designs, but certainly provides nearly optimal designs and show the major tendencies of the trade-off between the two objectives. Moreover, because the results are highly dependent on the room characteristics, panels parameters and climatic data, a complete optimization procedure would not be particularly relevant since any change of those parameters would create slight variations of the optimal designs.

5.2 Results and discussion of the multiobjective optimization

Figure 9 shows the Pareto fronts obtained for each value of the total length $L_{p,tot}$. For a color version of the figure, the reader is referred to the numerical version of the paper. First, Fig. 9a gives an example of a single Pareto front for $L_{p,tot} = 4m$. Each point represents one combination of a panel arrangement and fluid inlet temperature. The color indicates the fluid inlet temperature that was used in each case. Two different points may thus refer to the same panel arrangement, but with a different temperature. The Pareto curve obtained is smooth, but it should be noted that the fluid inlet temperature has an important influence on GDQ that increases rapidly when the temperature drops. The trade-off behaviour between the two objectives is such that a considerable reduction of the comfort is needed to achieve only a small energy consumption reduction.

Fig. 9b gives the disctinct pareto fronts for the three values of the total panel length: $L_{p,tot} = [2, 4, 6]\,m$. For clarity, $L_{p,tot} = 4m$ results are given as a tendency curve (i.e., solid line). This figure shows that there is generally no important difference in the optimal GDQ and Q'_{out} results that can be achieved for the different total panel length tested. This means that it is possible to achieve any Pareto optimal or nearly optimal combination of comfort and energy consumption with any total panel length. For example, well chosen $L_{p,tot} = 2m$ panel arrangements can allow the best comfort (but the highest energy consumption). They can also be the most energy efficient for GDQ over 10%, which could be exploited in applications such as garages or workshops, where some comfort can be sacrificed for energy cost reduction. However, small surface panel arrangements require high fluid inlet temperature.

Nevertheless, in most cases, optimal comfort will be desired, so an analysis of the zone within $GDQ < 5\%$ is suitable. Fig. 10 shows the Pareto front obtained in that region. For a color version of the figure, the reader is referred to the numerical version of the paper. Note that Fig. 10 is not a zoom of Fig 9b, because it does not show three different Pareto fronts, but a single front computed from all the simulated designs, regardless of their total panel length. However, markers indicating the total panel length of each design are still used for analysis. The diversity of the markers composing the Pareto front shows here again that Pareto optimal solutions can be found no matter what total panel length is chosen. As a comparison, the black dots in the figure show the performance of the eight panel arrangements (I to VIII) that were introduced in §4.2, at their respective optimal comfort fluid inlet temperature $T_{f,in,opt}$.

Also, three specially noticeable panel arrangements, one for each value of $L_{p,tot}$, are identified and schematically illustrated at the right of the figure. The first panel arrangement is for $L_{p,tot} = 2m$ and is the most comfortable design that was achieved among all the simulations. As mentioned in §4.1, the short ceiling panel near the window helps counterbalancing the cold effect of the façade to create a comfortable environment. The second panel arrangement is for $L_{p,tot} = 4m$, and is found on three points of the Pareto front, with $T_{f,in} = [301.5, 302, 302.5]K$. These points are remarkable by the fact that they are breakpoints on the Pareto curve and could thus be seen as slightly more optimal than the others. This is also the case with the last panel arrangement shown, that is for $L_{p,tot} = 6m$ and is found on two points of the Pareto front, with $T_{f,in} = [299.5, 300]$. This phenomenon confirms that some panel arrangements perform better than others, and that a family of Pareto optimal solutions can be achieved with these designs just by changing the fluid inlet temperature. In a general trend, results indicated that using a small panel near the window helps reaching a good level of comfort while a lower wall panel is especially efficient for reaching low energy consumption and minimizing the inlet fluid temperature needed.

Finally, it is important to note in Figs. 9 and 10 that the optimal fluid inlet temperatures can vary considerably depending on the total panel length imposed. It is generally much higher for the $L_{p,tot} = 2m$ designs. Although not included in the present analysis, this consideration could be important in a global system energy analysis, where the cost of the heat lost will be greater if the fluid temperature is higher. If the thermal energy is provided by a heat pump, the C.O.P. will also be lower, and the effective energy consumption higher. Furthermore, if the water is supplied by a renewable energy source like thermal solar collectors, it might be impossible to reach high temperatures without incurring a cost for reheating.

6. Conclusions

This paper showed a novel numerical method for evaluating the comfort and energy consumption of heating ceiling and wall hydronic radiant panels. The coupled CFD-SARPM method considered the real topology of the room and the radiant panel, as well as the airflow and radiant field characteristics. This allowed realizing a Pareto multiobjective optimization of the panels dimension and position, and fluid inlet temperature. The results yield the following conclusions:

- The comfort and energy consumption are contradictory objectives, which means that there is no single optimal design but a family of Pareto optimal designs that are good trade-offs between the two objectives.
- The thermal comfort is more sensitive than the energy consumption to the design variables. It is thus possible and appropriate to look for a solution that is close to minimizing the GDQ, without penalizing much on Q'_{out}.
- It is possible, even with small panel surface, to reach a great variety of Pareto optimal designs. However, this requires a warmer fluid than for larger surface panel area, which could increase the heat lost and reduce the heat pump performance for those designs.
- The fluid inlet temperature has a greater influence on GDQ and Q'_{out} than the panel arrangement. Thus, the greatest challenge for the optimization of these systems will be in the temperature control. Cost and technical considerations might thus be preponderant in the choice of the panels dimension and position.

Acknowledgements

This work was supported by the Natural Sciences and Engineering Research Council of Canada (NSERC). Tye-Gingras' work was also supported by the Fonds québécois de la recherche sur la nature et les technologies (FQRNT).

References

[1] McGraw Hill Construction, *Products and Player's Trend*. 2008.
[2] J. Miriel, L. Serres, and A. Trombe, « Radiant ceiling panel heating-cooling systems: experimental and simulated study of the performances, thermal comfort and energy consumptions », *Applied Thermal Engineering*, vol. 22, nº. 16, p. 1861-1873, nov. 2002.
[3] N. Djuric, V. Novakovic, J. Holst, and Z. Mitrovic, « Optimization of energy consumption in buildings with hydronic heating systems considering thermal comfort by use of computer-based tools », *Energy and Buildings*, vol. 39, nº. 4, p. 471-477, april 2007.
[4] Z. Tian and J. A. Love, « A field study of occupant thermal comfort and thermal environments with radiant slab cooling », *Building and Environment*, vol. 43, nº. 10, p. 1658-1670, oct. 2008.
[5] P. O. Fanger, *Thermal Comfort; Analysis and Applications in Environmental Engineering*. McGraw-Hill, 1970.
[6] T. Imanari, T. Omori, and K. Bogaki, « Thermal comfort and energy consumption of the radiant ceiling panel system.: Comparison with the conventional all-air system », *Energy and Buildings*, vol. 30, nº. 2, p. 167-175, 1999.
[7] N. Ghaddar, M. Salam, and K. Ghali, « Steady Thermal Comfort by Radiant Heat Transfer: The Impact of the Heater Position », *Heat Transfer Engineering*, vol. 27, nº. 7, p. 29-40, 2006.
[8] T. Catalina, J. Virgone, and F. Kuznik, « Evaluation of thermal comfort using combined CFD and experimentation study in a test room equipped with a cooling ceiling », *Building and Environment*, vol. 44, nº. 8, p. 1740-1750, 2009.
[9] J. A. Myhren and S. Holmberg, « Flow patterns and thermal comfort in a room with panel, floor and wall heating », *Energy & Buildings*, vol. 40, nº. 4, p. 524-36, 2008.
[10] N. F. Diaz, J. Lebrun, and P. Andre, « Experimental study and modeling of cooling ceiling systems using steady-state analysis », *International Journal of Refrigeration*, vol. 33, nº. 4, p. 793-805, june 2010.
[11] R. K. Strand and K. T. Baumgartner, « Modeling radiant heating and cooling systems: integration with a whole-building simulation program », *Energy and Buildings*, vol. 37, nº. 4, p. 389-397, 2005.
[12] ASHRAE Handbook 2008:, *HVAC Systems and Equipment*. American Society of Refrigerating and Air-Conditioning Engineers. Atlanta, Georgia.
[13] J.-W. Jeong and S. A. Mumma, « Ceiling radiant cooling panel capacity enhanced

by mixed convection in mechanically ventilated spaces », *Applied Thermal Engineering*, vol. 23, n°. 18, p. 2293-2306, 2003.
[14] C.L. Conroy and S. A. Mumma, « Ceiling Radiant Cooling Panels as a Viable Distributed Parallel Sensible Cooling Technology Integrated with Dedicated Outdoor Air Systems », *ASHRAE Transactions*, vol. 107, n°. 1, p. 578-585, 2001.
[15] J.-W. Jeong and S. A. Mumma, « Practical cooling capacity estimation model for a suspended metal ceiling radiant cooling panel », *Building and Environment*, vol. 42, n°. 9, p. 3176-3185, 2007.
[16] H. Hanibuchi and S. Hokoi, « Simplified method of estimating efficiency of radiant and convective heating systems », vol. 106, n°. 1, p. 487-494, 2000.
[17] N. Fonseca Díaz, « Experimental study of hydronic panels system and its environment », *Energy Conversion and Management*, vol. In Press, Corrected Proof.
[18] N. Fonseca Diaz, « Modeling of a hydronic ceiling system and its environment as energetic auditing tool », *Applied Energy*, vol. 88, n°. 3, p. 636-649, 2011.
[19] M. Tye-Gingras and L. Gosselin, « Investigation on heat transfer modeling assumptions for radiant panels with serpentine layout », *Energy and Buildings*, vol. 43, n°. 7, p. 1598-1608, 2011.
[20] F. C. McQuiston, J. D. Parker, and J. D. Spitler, *Heating, Ventilation and Air Conditioning Analysis and Design*, 6e éd. John Wiley & Sons, Inc., 2005.
[21] M. Tye-Gingras and L. Gosselin, « Optimisation morphologique de panneaux radiants hydroniques », in *Actes du Xème Colloque Interuniversitaire Franco-québécois sur la Thermique des Systèmes*, Saguenay, 2011.
[22] F. R. Menter, « Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications », *AIAA Journal*, vol. 32, n°. 8, p. 1598-1605, august. 1994.
[23] F. R. Menter, M. Kuntz, and R. Langtry, « Ten Years of Experience with the SST Turbulence Model », In K. Hanjalic, Y. Nagano, and M. Tummers, editors, *Turbulence, Heat and Mass Transfer 4*, pages 625-632. Begell House Inc., 2003.
[24] W. Xu and O. Chen, « Numerical Simulation of Airflow in a Room with Differentially Heated Vertical Walls », *ASHRAE Transactions*, vol. 104, n°. 1, 1998.
[25] E. H. Chui and G. D. Raithby, « Computation of radiant heat transfer on a nonorthogonal mesh using the finite-volume method », *Numerical Heat Transfer, Part B (Fundamentals)*, vol. 23, n°. 3, p. 269-88, april 1993.
[26] M. F. Modest, Radiative Heat Transfer, 2nd Edition. Academic Press, Elsevier, 2003.
[27] D. A. Olson, L. R. Glicksman, and H. M. Ferm, « Steady-state convection in empty and partitioned enclosures at high Rayleigh numbers », *J. Heat Transfer, ASME*, vol. 112, n°. 3, p. 640-647, 1990.
[28] ISO7730:1994, *Moderate Thermal Environments—Determination ofthe PMVand PPD Indices and Specification of the Conditions for Thermal Comfort, revised version*. Geneva: International Organization for Standardization, 1994.
[29] K. Deb, A. Pratap, S. Agarwal, and T. Meyarivan, « A fast and elitist multiobjective genetic algorithm: NSGA-II », *IEEE Transactions on Evolutionary Computation*, vol. 6, n°. 2, p. 182-97, april 2002.
[30] D. A. Van Veldhuizen and G. B. Lamont, « Multiobjective evolutionary

algorithms: analyzing the state-of-the-art », *Evolutionary Computation*, vol. 8, nº. 2, p. 125-47, Summer. 2000.
[31] K. Deb, *Multiobjective Optimization Using Evolutionary Algorithms*. Chichester, UK: Wiley, 2001.
[32] C. M. Fonseca and P. J. Fleming, « An overview of evolutionary algorithms in multiobjective optimization », *Evol. Comput.*, vol. 3, nº. 1, p. 1–16, march 1995.
[33] L. Gosselin, M. Tye-Gingras, and F. Mathieu-Potvin, « Review of utilization of genetic algorithms in heat transfer problems », International Journal of Heat and Mass Transfer, vol. 52, no. 9-10, p. 2169-2188, 2009.

Figure captions

Figure 1	Representation of the zone with one façade wall with a window and the radiant panels on the ceiling and right wall.
Figure 2	Schematic representation and stepwise description of the iterative resolution procedure.
Figure 3	Comparison of: a) the horizontal temperature profile near the hot wall at $y = H_r/2$ with $T_H = 29.3\,^{o}C$ and $T_C = 10.9\,^{o}C$, b) the vertical temperature profile at $x = L_r/2$ with $T_H = 35.3\,^{o}C$ and $T_C = 19.9\,^{o}C$.
Figure 4	Schematic representation of a serpentine hydronic radiant panel.
Figure 5	Various temperature and comfort results obtained with a centered 2m long ceiling panel.
Figure 6	Schematic representation of the eight panel arrangements simulated.
Figure 7	Influence of $T_{f,in}$ on GDQ and Q'_{out} for the eight panel arrangements simulated.
Figure 8	Effect of mass flow rate change and fluid tube inlet permutation on the GDQ and Q'_{out} for the panel arrangements I and VI.
Figure 9	Pareto fronts showing the trade-off between GDQ and Q'_{out} for a) $L_{p,tot} = 4m$ and b) $L_{p,tot} = 2, 4, 6\,m$.
Figure 10	Combined Pareto front of all the designs tested for the optimal zone within $GDQ < 5\%$.