Journal of Southeast University (English Edition) Vol. 26, No. 2, pp. 364 – 367 June 2010 ISSN 1003—7985

Cooling performance analysis of radiant panel at different positions

Gao Zhihong Liu Xiaohua Jiang Yi

(School of Architecture, Tsinghua University, Beijing 100084, China)

Abstract: The influence of the panel position on the cooling performance of a radiant panel is analyzed. The coupled simulation of convection and radiation is set up by a computational fluid dynamics (CFD) method. The simulations with different panel positions and different indoor heat sources are used to calculate the cooling capacity of the radiant panel and the indoor thermal environment. The simulation results are in good agreement with the experimental results. The results show that when the indoor heat source temperature is low, the convective heat flux is the main influence factor of the cooling capacity and the radiant panel should be placed on the wall or on the ceiling. Otherwise, when the indoor heat source temperature is high, the radiation heat flux is the main factor and the radiant panel should be placed as near to the heat sources as possible.

Key words: radiant panel; panel position; cooling capacity

Radiant panels have received increasing attention in recent years, especially in commercial buildings. It is often claimed that the radiant panels can improve thermal comfort and increase energy efficiency[1–2]. Radiant cooling ceilings and radiant heating floors are the most popular radiant panels, but there is little study and analysis on how to design and choose the panel position in different indoor environments.

Kim et al. [3–4] reported a measurement and simulation study on the indoor thermal environment of office space controlled by a radiant panel system installed in the vertical wall. The results show that with the same amount of cooling load, the cooling system using a radiant panel has better cooling efficiency than the all-air cooling system. Wang et al. [5] described a simulation of a radiant cooling system with an air-handling unit for dehumidification, which can establish guidelines for designing a low-energy system and for targeting the average PMV and the exergies of the chilled water. The results show that when the load rate of the cooling loads change from 100% to 57% and 27%, the exergies decrease by 47% and 67%, respectively. When the radiant panel area and the average PMV increase, the exergy of the chilled water decreases.

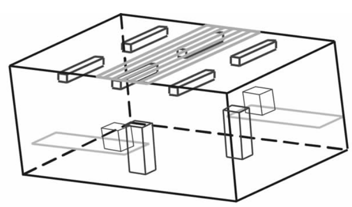
The total heat removal of the radiant panel contains three parts: long-wave radiation, convection and short-wave radiation from solar power or illumination. The short-wave radiation is not considered in this research. The convection heat is influenced by the air temperature and the convective heat transfer coefficient. The radiation heat is impacted by the heat source temperature and the view factor between the panel and the heat source. The panel position of the radiant panel can influence the convective heat transfer coefficient and the view factor; thus, changing the panel position will have an impact on the cooling capacity of the radiant panel. In this paper, the influence of the panel position on the cooling performance is examined from a practical point of view.

1 Case Study

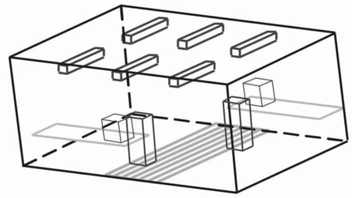
The simulation of the indoor air flow is conducted by a convection and radiation coupled simulation based on a standard k-ε model. In this research, the humidity transportation equation is not considered. The heat transmitted through the outer wall is computed at a given air temperature.

A feedback system is added to the simulation. During the simulation, the feedback system modifies the area conditions of the radiant panel to attain the same operation temperature (OT). The target operation temperature is 26 ℃. Subsequently, the system output is modified to keep the operation temperature at the target value. Using this method, the precise cooling capacity of the panel can be analyzed at the same sensation level.

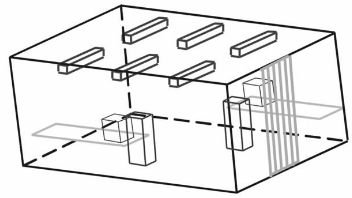
In this study, the radiant panels are placed on the ceiling, the floor and the wall, respectively, as shown in Fig 1. And several room models (office room model, classroom model,



(a)



(b)



(c)

Fig. 1 Schematic representation of radiant panel at different positions. (a) On the ceiling; (b) On the floor; (c) On the wall

Received 2009-11-26.

Biographies: Gao Zhihong (1984—), female, graduate; Jiang Yi (corresponding author), male, doctor, professor, jiangyi@ tsinghua. edu. cn.

Foundation item: The National Natural Science Foundation of China (No. 50778094).

Citation: Gao Zhihong, Liu Xiaohua, Jiang Yi. Cooling performance analysis of radiant panel at different positions[J]. Journal of Southeast University (English Edition), 2010, 26(2): 364 – 357.

test room model and modified office room model) are simulated, as shown in Fig. 2. The dimensions of the room are 5.16 m × 3.65 m × 2.43 m. In the test room model, there is only one electrical heater which is used as the testing heat source. The total cooling loads in the office room, classroom and test room models are shown in Tab. 1. All the walls are heat-adiabatic and there is no heat transferred from outside. In the modified office room model, there is an exterior wall in the west side with a heat transfer coefficient of 0.7 W/(m·K) and the outside temperature, which is used to calculate the heat transmission, is 34 ℃. Thus, the cooling load in the modified office room model changes at different surface temperatures of the radiant panel.

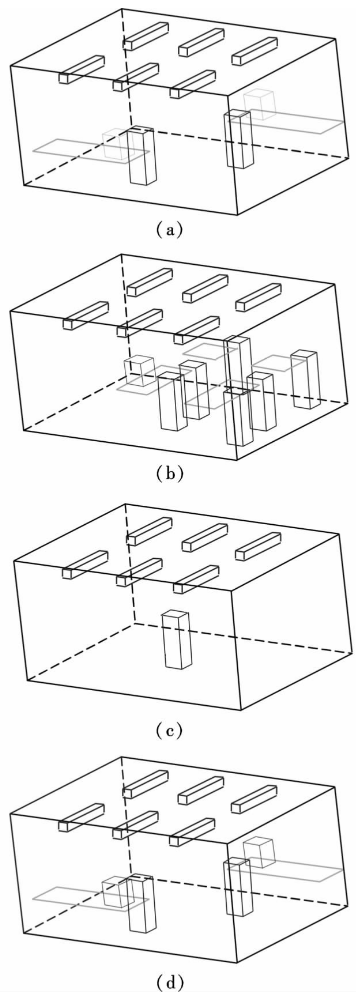


Fig. 2 Simulation models. (a) Office room model; (b) Classroom model; (c) Test room model; (d) Modified office room model

Tab. 1 Cooling loads of different models W

Mode	Human	Lighting	Computer	Heater	Total
Office room	150	204	200		554
Classroom	450	204	100		754
Test room		204		1 000	1 204

2 Analysis

2.1 Thermal comfort

The air temperature (AT) and the mean radiant temperature (MRT) are two main factors influencing the indoor thermal comfort of a human body. According to ANSI/ASHRAE 55—1992, the operation temperature is used to evaluate the thermal comfort. That is,

$$t_{\mathrm{o}} = \frac{h_{\mathrm{r}} t_{\mathrm{r}} + h_{\mathrm{c}} t_{\mathrm{a}}}{h_{\mathrm{r}} + h_{\mathrm{c}}} \tag{1}$$

$$t_{\mathrm{r}} = \left(\sum_{i=1}^{n} F_i T_i^4 \right)^{1/4} - 273 \tag{2}$$

where t_{a} is the air temperature; h_{r} and h_{c} are the radiant and convective heat transfer coefficients, respectively; t_{r} is the mean radiant temperature; T_i is the temperature of the i-th object.

2.2 Calculation conditions for simulation

$$q_{\mathrm{tot}} = q_{\mathrm{c}} + q_{\mathrm{r}} + q_{\mathrm{solar}} \tag{3}$$

$$q_{\mathrm{c}} = h_{\mathrm{c}} (t_{\mathrm{a}} - t_{\mathrm{p}}) \tag{4}$$

$$q_{\mathrm{r}} = \sum_{j=1}^{n} F_{\mathrm{p}j} J_j - J_{\mathrm{p}} \tag{5}$$

$$\frac{t_{\mathrm{p}} - t_{\mathrm{m}}}{R} = q_{\mathrm{tot}} \tag{6}$$

$$\frac{G_{\mathrm{w}} c_{\mathrm{p}} (t_{\mathrm{out}} - t_{\mathrm{in}})}{A} = q_{\mathrm{tot}} \tag{7}$$

where q_{tot}, q_{c}, q_{r}, q_{solar} are the total heat, the convective heat, the radiation heat and the solar radiation heat of the radiant cooling panel, respectively; J_{p} is the radiation node of the radiant cooling panel; R and A are the thermal resistance and the area of the radiant panel, respectively; t_{out} and t_{in} are the outlet and inlet temperatures of the cold water, respectively; t_{p} is the average temperature of the radiant panel; G_{w} is the mass flow rate of the cold water; c_{p} is the specific heat of the cold water.

When the surface temperature of the radiant cooling panel is used as a boundary condition, we can calculate the convective and radiation heat by the CFD simulation. From Eq. (6), the water temperature can be calculated by a one-dimensional heat conduction calculation. Assuming the temperature difference between the outlet water and the inlet water to be 3 ℃, we can obtain the flow rate of the water, G_{w}. From Eq. (7), we can obtain the inlet temperature of the chilled water supplied to the radiant panels. According to the set temperature difference, the outlet temperature can be calculated.

3 Results and Discussion

A chamber experiment is conducted to verify the simulation results, and the photo of the used radiant panel is shown in Fig. 3. In the experiment, the indoor heat sources are computers and occupants. The simulation and measurement results of the vertical air temperature distribution are shown in Fig. 4. It can be found that the CFD model provides sufficient accuracy as a tool for estimating the indoor thermal environment.

Fig. 3 Radiant panel in experiment

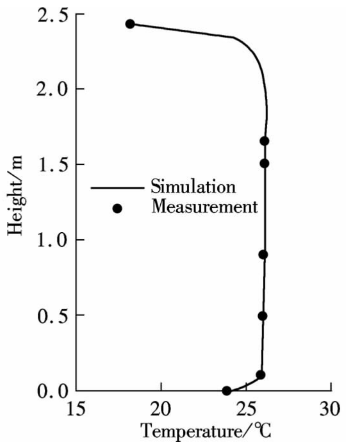


Fig. 4 Simulation and measurement results of vertical air temperature distribution

3.1 Cooling capacity

The relationship between the panel position and the cooling capacity is shown in Fig. 5. The cooling capacity can be calculated by the heat flux on the surface of the radiation cooling panel. As for the office room and classroom models, because of the low convective heat transfer coefficient, the convective heat flux and the cooling capacity of the radiant panel placed on the floor are lower than those of the radiant panels placed on the wall and on the ceiling. However, the radiation heat fluxes of all the radiant panels are similar. With the increase in the heat source temperature, the radiation heat flux and the cooling capacity increase. Thus, the radiant panel should be put near to the heat sources, which can make the radiant heat transfer enhanced. As a result, when the heat source temperature is low, the convective heat flux is the main influence factor of the cooling capacity, and the radiant panel should be placed on the wall or on the ceiling; otherwise, the radiation heat flux is the main factor and the radiant panel should be placed near the heat sources.

3.2 Comparison of thermal sensation

The mean radiant temperature and the operation temperature of a human body with different positions of the radiant panels are shown in Fig. 6. When the radiant panels are placed on the ceiling or on the wall, the MRT of the human body is about 26 ℃, which is similar to the air temperature. When the radiant panel is placed on the floor, the MRT of the human body is about 1 to 2 ℃ lower than those when the radiant panels are placed on the ceiling and on the wall. That is to say, when the radiant panel is placed on the floor,

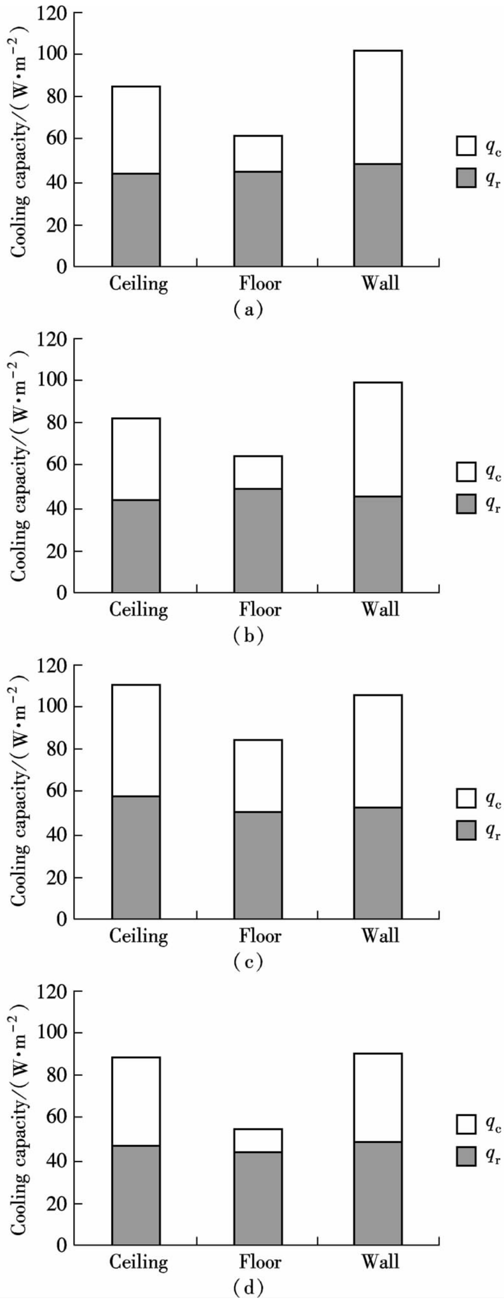


Fig. 5 Cooling capacities of radiant panels. (a) In office room model with average heat source temperature of 32 ℃; (b) In classroom model with average heat source temperature of 33 ℃; (c) In test room model with average heat source temperature of 44 ℃; (d) In modified office room model with average heat source temperature of 32 ℃

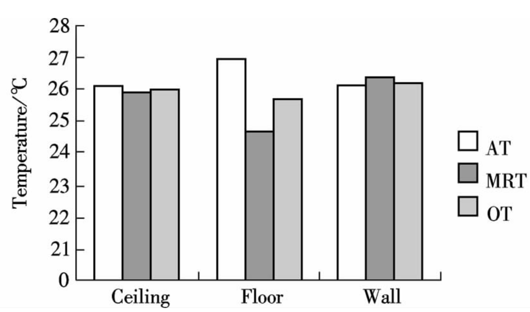


Fig. 6 MRT and OT of human body

the air temperature can be 1 to 2 ℃ higher than that when it is placed on the ceiling or on the wall.

3.3 Heat transfer through exterior wall

There are three factors influencing the heat transmission from the exterior wall: the panel position, the panel area and the surface temperature of the radiant panel. In the modified office room model, when the surface temperature of the radiant panel decreases from 19 to 17 ℃, in order to maintain the same operation temperature, the panel area decreases accordingly. The lower surface temperature can lead to greater heat transmission, but the lower panel area can result in lower heat transmission. Fig. 7 shows that when the surface temperature and panel area decrease, the heat transmission decrease accordingly. It means that the panel area plays a more important role in the heat transmission from the exterior wall. Furthermore, when the radiant panel is placed on the floor, the heat transmission from the exterior wall is the lowest.

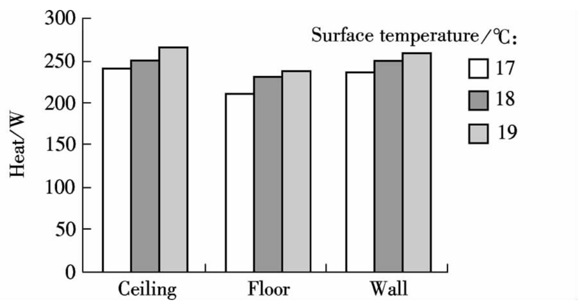


Fig. 7 Heat transferred from exterior wall

4 Conclusions

By using a numerical simulation tool, we investigate the performance of radiant panels at different positions. The results can be summarized as follows:

1) A CFD simulation and a convection and radiation heat transfer simulation for analyzing the radiant panels are presented in this study. The required cooling capacities at different panel positions are quantitatively evaluated.

2) The cooling capacity of the radiant panel varies with the indoor heat source. When the indoor heat source temperature is low, the convective heat flux is the main influence factor of the cooling capacity, and the radiant panel should be placed on the wall or on the ceiling.

3) When the indoor heat source temperature is high, the radiation heat flux is the main factor and the radiant panel should be placed as near to the heat sources as possible.

4) When the radiant panel is placed on the floor, the heat transmission from the exterior wall is the lowest.

References

[1] Bean R, Olesen W, Kim W. History of radiant heating and cooling systems [J]. *ASHRAE Journal*, 2010, **52**(2): 50 − 55.

[2] Imanari T, Omori T, Bogaki K. Thermal comfort and energy consumption of the radiant ceiling panel system: comparison with the conventional all-air system [J]. *Energy and Buildings*, 1999, **30**(2): 167 − 175.

[3] Kim T, Kato S, Murakami S, et al. Study on indoor thermal environment of office space controlled by radiant panel system using field measurement and numerical simulation [J]. *Building and Environment*, 2005, **40**(3): 301 − 310.

[4] Kim T, Kato S, Murakami S. Indoor cooling/heating load analysis based on coupled simulation of convection, radiation and HVAC control [J]. *Building and Environment*, 2001, **36**(7): 901 − 908.

[5] Wang S, Morimoto M, Soeda H, et al. Evaluating the low exergy of chilled water in a radiant cooling system [J]. *Energy and Buildings*, 2008, **40**(10): 1856 − 1865.

不同位置辐射板的供冷性能分析

高志宏　刘晓华　江　亿

(清华大学建筑学院,北京 100084)

摘要:分析了辐射板布置位置对供冷性能的影响,建立了一种对流与辐射耦合的 CFD 模拟模型. 通过模拟计算,得到不同辐射板位置和室内热源状况下辐射板供冷能力及室内热环境的分布状况;该模拟结果与真实平台实验结果基本吻合. 模拟结果表明:当室内热源温度较低时,对流换热量为主导因素,因此应将辐射板放置于侧墙或天花板处;当室内热源温度较高时,辐射板与热源的辐射换热量增大,辐射换热量为主导因素,因此应将辐射板布置在靠近高温热源的位置.

关键词:辐射板;辐射板位置;供冷能力

中图分类号:TU831