Radiant Ceiling Panel (RCP) for heating/cooling application
Part 3. A numerical comparison between flat and a novel curved surface RCP

正 会 員 〇Ahmed A.Serageldin（北海道大学） 学生会員 叶 敏之（北海道大学）
正 会 員 長野 克則（北海道大学）
Ahmed A.Serageldin*1 Minzhi YE*1 Katsunori NAGANO*1
*1 Hokkaido University

The comparison between the thermal performance and thermal comfort conditions of the flat surface radiant panel and a novel concave curved radiant panel under the same operating conditions is carried out by numerical investigation. Different curvature cord length to curvature radius ratios (L/R) from 0 to 2 are examined. This study includes a comprehensive heat transfer analysis as well as an exhausted thermal comfort assessment is carried out for each case. Three dimensional CFD model is developed and validated against the experimental results in the literature, the model shows a good agreement with acceptable accuracy.

1. Introduction

Recently, the application of the Radiant Heating/Cooling (RHC) system is widespread worldwide because of four main characteristics: its tendency of low energy consumption [1,2], an acceptable thermal comfort level [2,3], low risk of condensation [4], and better indoor air quality [5]. Seeking more energy saving towards a nearly zero energy heating/cooling systems, So the current RHC technology needs more research and developments.

In this study, a novel curved surface radiant ceiling panel (RCP) is numerically investigated. A three-dimensional CFD simulations are carried out by ANSYS FLUENT environment to analysis the heat transfer characteristics and thermal comfort aspects inside a single room equipped with an innovated RCP. Different curvature cord length-to-curvature radius (L/R) are compared with the customary flat surface panel.

2. CFD model Set-up

The three-dimensional finite volume model is developed by ANSYS FLUENT software to solve the coupling between fluid flow and heat transfer inside a full-scale room under steady-state conditions. The dummy surfaces ejected the heat energy transferred by convection to the indoor air and by radiation to the other surfaces of walls, roof, ceiling, and panels surfaces. The surface-to-surface radiation model calculates the radiation exchange in an enclosure with gray-diffuse surfaces. This model relies on calculating the view factor between surfaces that accounting the surface size, separation distance, and orientation.

2.1. Geometry

The geometry consists of the full-scale room with a height, width, and depth of 4 m, 4m, and 2.9 m, which is equipped with a suspended metal RCP at 0.3 m beneath the ceiling level. The RCP surface area is 12.96 m^2, with 3.04 m^2 free space between the panel and walls. Two different panels are used; one with a flat surface and the other with a curved surface. The curved panel that has the same surface area, but with segmented curves, each segmented curve has a length of 0.1 m. The curvature cord length ($\boldsymbol{l}$) and radius ($\boldsymbol{r}$) are adopted to achieve the highest convective heat transfer, so the $\boldsymbol{l/r}$ ratio is studied in the span between 1.99 and 0.33 as shown as listed in Table 1. Also, the room is heated internally by twelve cylindrical dummies, which are mimicking the human bodies with a diameter of 0.3 m, and a length of 1.1 m, as shown in Fig. 1.

2.2. The meshing of the geometry

The geometry is discretized by using a tetrahedral element with adaptable element size, the size is coarsened in the middle area and refined close to the surfaces of CRP and dummy cylinders.

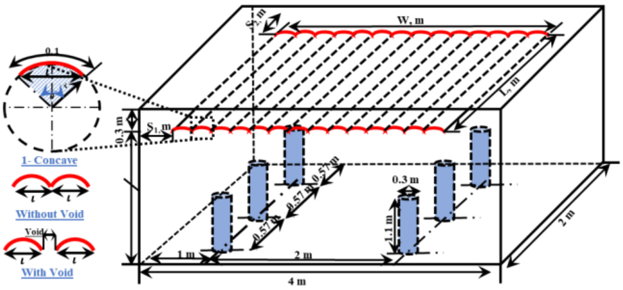

Fig. 1 Geometric dimensions used in this study

Table 1 The dimensions of cases used in this study

r	ι	ι/r	θ	Void	$S1$	W	Ap/A
0.03	0.0597	2	95.49	-	0.7715	2.457	0.55
0.03	0.0597	2	95.49	0.029	0.2000	3.600	0.81
0.09	0.0950	1.1	31.83	-	0.2900	3.420	0.77
0.2	0.0990	0.5	14.32	-	0.2180	3.564	0.80

The inflation layers are adopted near the walls, dummy cylinder, and the CRP surfaces, the first layer thickness is 0.001 m, and the total thickness is 3 mm with a growth rate of 1.2, as shown in Fig. 2.

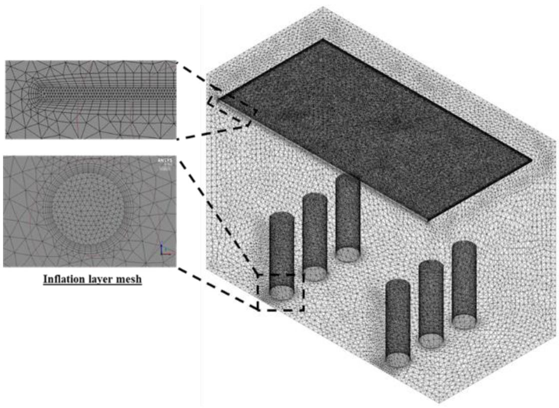

Fig.2 Grid of the simulation domain and the inflation layers

2.3. Physical model

ANSYS FLUENT multi-physics solver was used to solve the coupling between fluid flow and heat transfer inside a closed enclosure. The realizable k-epsilon (k-ε) turbulence model is used to simulate the turbulent kinetic energy, k, and the turbulent dissipation rate, ε, and study the turbulence properties of the flow inside the space. The surface-to-surface (S2S) radiation model is used to calculate the radiation heat exchange between two surfaces.

2.4. Boundary conditions and solver schemes

The boundary conditions are assigned to the walls, RCP, and dummy cylinders surfaces are selected similarly to the experimental work conducted by Shin et al. The boundary conditions are summarized as shown in Fig. 3 and Table 2. The heat flux assigned on the outer surface of the dummy cylinder is mimicking the heat generated from the human body.

Table 2 Boundary conditions and emissivity of each surface

Boundary	Condition	T	Q,W/m^2	ε(-)
Room_Wall	Adiabatic	-	0	0.82
Panel _Surface	Ts=constant	15.83	-	0.92
Cylinder_Outer_Wall	qs=constant	-	113	0.92

The room is well insulated, thus the adiabatic boundary conditions are appointed to the room walls, also the panel surface is assumed to be at a uniform and isothermal temperature. On the other hand, the heat energy outgoes from the dummy surface is ejected from the peripheral surface, so the upper surface of the dummy cylinder is considered as an adiabatic boundary.

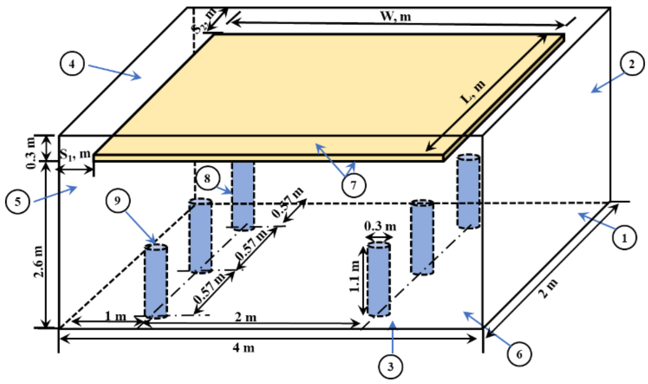

Fig. 3 Boundary conditions used in this study

3. Results and discussions

The thermal performance of radiant ceiling panel used for cooling applications was numerically studied by three-dimensional finite volume method (FVM). A comparison between the flate surface panel and a novel curved surface was carried out. Firstly, the CFD simulation is validated with the experimental results summarized in Ref.[2]. Figure 4 shows that the CFD model had a good agreement with the experimental results with average error percentage of 1.1 %.

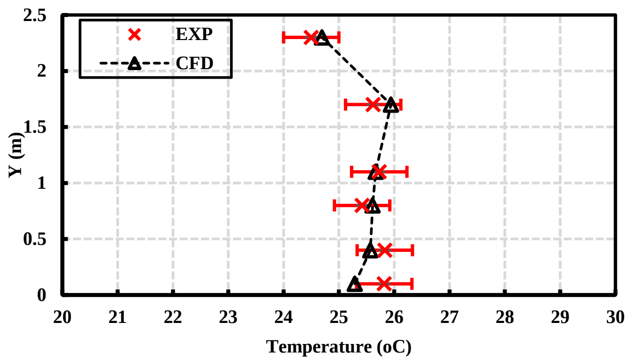

Fig. 4 Comparison between experimental and CFD results

3.1. Heat transfer analysis results

3.1.1 Air Temperature profile

The air temperature distribution a long a vertical line drawn at the mid-plane is calculated and shown in Fig.5. So, as the coverage area increases from 55 % 81 % for L/R of 2, the average air temperature decreases by 2 °C from 26.41 °C to 24.41 °C. Also, the floor temperature decreases from by 1.62 °C from 27.18 °C to 25.56 °C which achieve more comfort satisfaction. The indoor air temperature contours drawn on a vertical plane at Z=1.7 m, and two planes crossing the dummy cylinders for each L/R value are shown in Fig. 6.

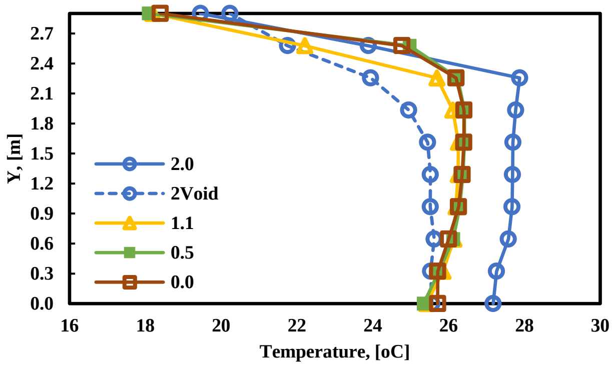

Fig.5 Vertical air temperature profile air temperature

Figure 7 shows the average air temperature, an uncooled surface temperatures (AUST) and operative temperature for different L/R ratio starts from 0 to 2. The AUST increases from 26.58 oC to 27.12 oC when L/R increases from 0 to 2, but with adding void to the ratio's L/R of 1.5 and 2, the AUST decreases to 26.54 oC and 26.78 oC, respectively.

3.1.2 Heat transfer coefficients

Figure 8a shows the individual values of Qc and Qr for

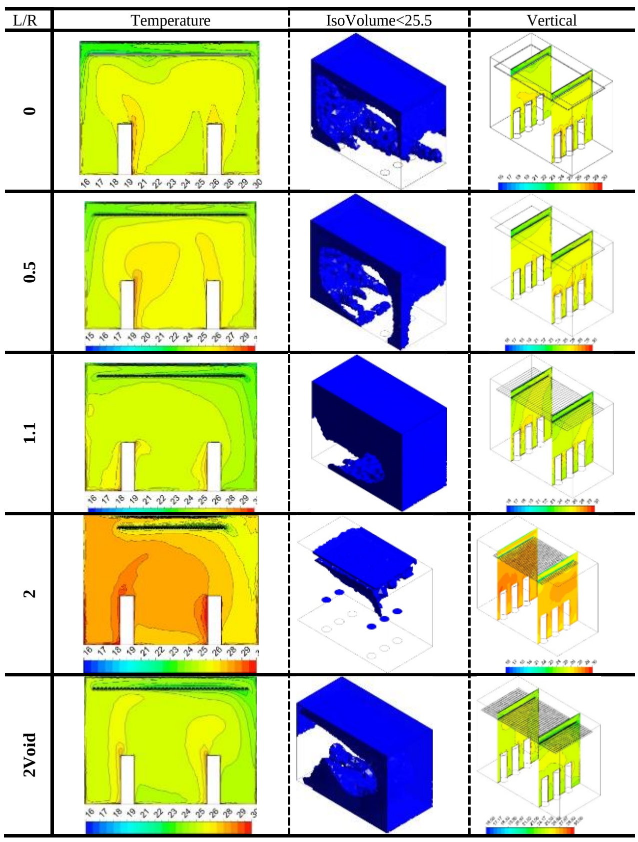

Fig. 6 Temperatures contours and IsoVolume in each case

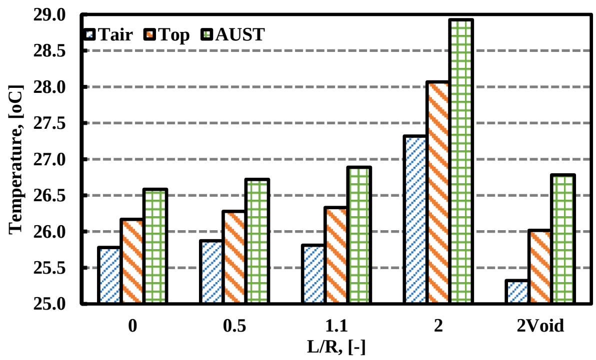

Fig. 7 Air, operative and AUST temperatures for different L/R

each case. The convection part Qc increases with increasing the L/R values from 0 to 2, it increased by 30 % from 220 W to 280 W, and the radiation part decreases from 482.7 W to 409.7 W, as well. Consequently, the convection and radiation heat transfer coefficients are calculated and summarized in Fig. 8b. Moreover, increasing the ratio L/R from 0 to 2, decreases the h_r by 32 % from 5.3 W/m^2.K to 3.6 W/m^2.K, and the h_c increases at the upper surface by 174 % from 1.02 W/m^2.K to 2.8 W/m^2.K as shown in Fig.8c. the range of -1~ 1.3, and average value of -0.26, while 75% of the space volume lower than -0.06.

3.1.3 Thermal Comfort Results

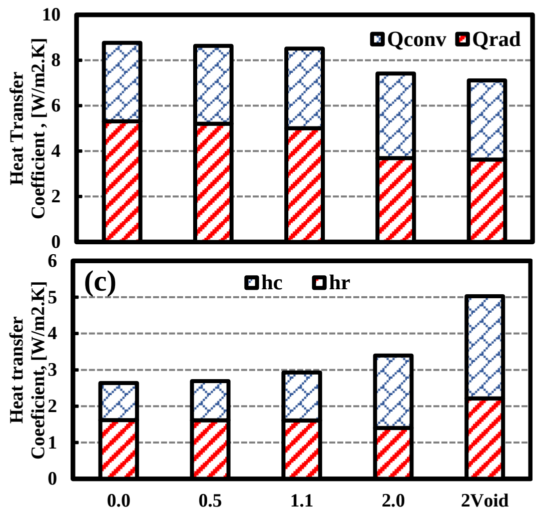

Fig8. heat transfer coefficient on a) the Bottom and b) Upper panel surfaces

For L/R of 1.5 and void, the minimum value is -0.09, maximum value is 0.47 and the outliner is 1.28 as well. Also, the median is increases to -0.2 and 75 % of the indoor air volume has a PMV value lower than -0.06 as shown in Fig.9.

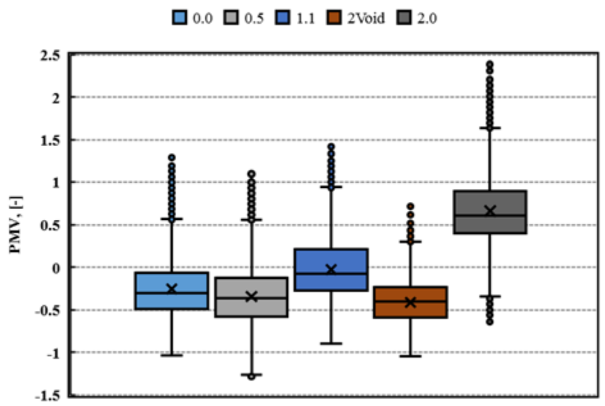

Fig.9 Boxplot of PMV index for each case

These results can be explained by contours the PMV and PPD indexes on the vertical plane as shown in Fig.10.

4. Conclusions

The results are concluded as follows:

1- The CFD model had a good agreement with the experimental results with average error percentage of 1.1 %.

2- As the coverage area increases from 55 % 81 % when L/R is 2, the average air temperature decreases by 2 °C from 26.41 °C to 24.41 °C

3- The ratio L/R of 1.5 with void distance shows the lowest operative temperature of 25.84 °C followed by the ratio L/R of 2 with a void space to 26.02 °C.

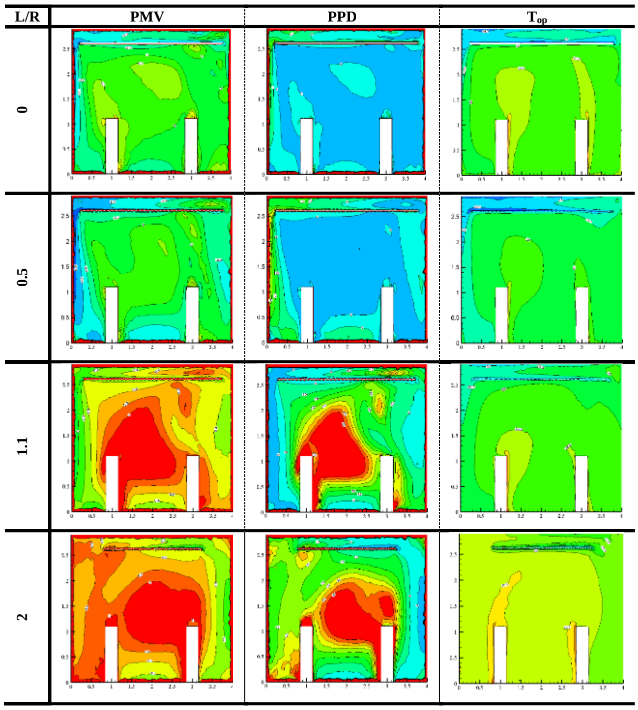

Fig. 10 PMV index, PPD index and Operative temperature.

4- Increasing the ratio L/R from 0 to 2, decreases the h_r by 32 % from 5.3 W/m^2.K to 3.6 W/m^2.K, and the h_c increases at the upper surface by 174 % from 1.02 W/m^2.K to 2.8 W/m^2.K

5- For flat surface panel, the PMV values in the range of -1~ 1.3, and average value of -0.26, while 75% of the space volume has a value of PMV lower than -0.06.

References

[1] Sastry G, Rumsey P. VAV vs. Radiant. ASHRAE J 2014;56:17–24.

[2] Imanari T, Omori T, Bogaki K. Thermal comfort and energy consumption of the radiant ceiling panel system. Energy Build 1999;30:167–75.

[3] Niu J, Kooi J v.d., Rhee H v.d. Energy saving possibilities with cooled-ceiling systems. Energy Build 1995;23:147–58.

[4] Lin B, Wang Z, Sun H, Zhu Y, Ouyang Q. Evaluation and comparison of thermal comfort of convective and radiant heating terminals in office buildings. Build Environ 2016;106:91–102.

[5] Stetiu C. Energy and peak power savings potential of radiant cooling systems in US commercial buildings. Energy Build 1999;30:127–38.