Radiant Ceiling Panel (RCP) for heating/cooling application
Part 2. Parametric study on thermal performance of RCP with a novel curved surface by numerical approach

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 thermal performance and thermal comfort conditions inside a room equipped with of a radiant ceiling with a concave segmented curved surface is carried out by numerical investigation where different panel arrangement is 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, the model shows a good agreement with acceptable accuracy. Also, better arrangement enhances the heat transfer inside the room.

1. Introduction

The radiant ceiling panel (RCP) is considered as a low energy heating and cooling system which enhances the thermal comfort of buildings[1]. So, a high supply water temperature could be circulate through RCP during cooling operation and a lower temperature during heating purposes[2,3]. Size and numbers of RCP panels as well as the arrangement have a significant impact on the indoor thermal comfort and energy consumption.[4]. So, this study investigates the impact of various panel orientation on the heat transfer and thermal comfort inside a laboratory environmental room cooled by suspended metal RCP with a concave curved segmented surface by experimental and numerical approaches.

2. Experimental Set-Up

The experiment set-up consists of twin identical environmental chambers with autonomous control. Each chamber has a dimension of length, width and height of 2.7 x 2.7 x 2 m^3. Also, the two chambers are sharing a wall with an opening of dimensions 1.3 m and 0.9 m, this opening is blocked by a double-glazing window. The right chamber is equipped with the radiant ceiling panels, While, the left chamber is mimicking the outdoor environment temperature in Summer and Winter seasons. Four panels with a length of 1.782 m and width 0.583 m and the void spaces between panels are 2 cm and 5 cm in Z and X directions are hanged at 19 cm beyond the ceiling as shown in Fig. 1. The room is heated internally by four cylindrical heat generating elements. The cylinder is made from aluminum sheet with a diameter of 25 cm and length 1.2 m and thickness of 1 mm, Also, it painted with a black color. The heat is generated by an incandescent bulb with a power of 100 W, the heat is radiated and convected from the outer surface of the cylinder to surroundings as shown in Fig. (2).

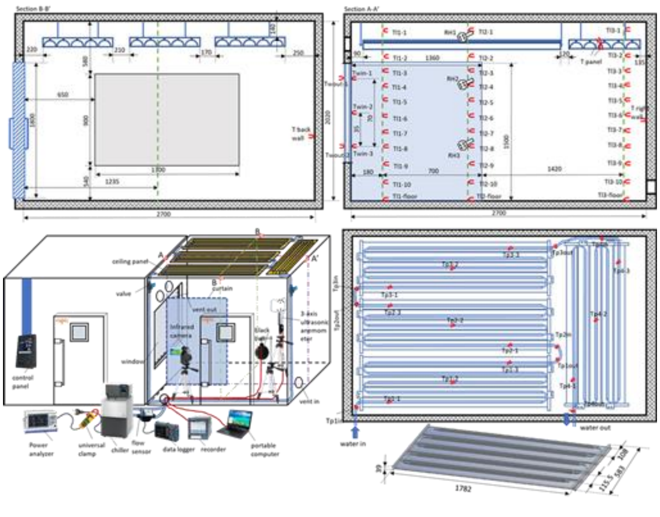

Fig. 1Experimental Set-Up schematic diagram

3. Experimental Procedure

The experiments are conducted according to ASHRAE standard, for heating and cooling proposes. The experiments procedure is explained as follows:

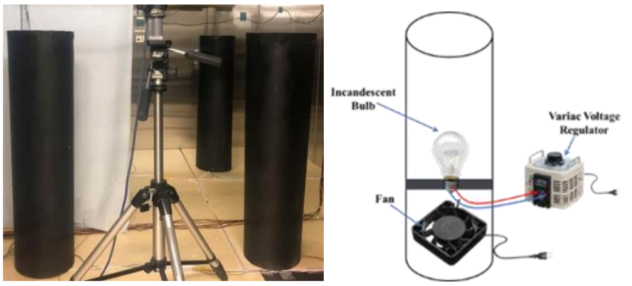

Fig. 2 Photo and schematic diagram of Internal load

a) The air temperature in the left chamber is adjusted at 40 ºC

b) Simultaneously, the incandescent lamps inside the internal heat generation dummies are switched on.

c) Then, the indoor air temperature of the right chamber is monitored until it reaches the steady-state temperature, which is around 30 oC±1 ºC.

d) Then, the chilled water is circulated through the panels tube with a predefined flow rate and temperature.

e) While, the measurements of temperatures, relative humidity, and air velocity are recorded every 1 second.

f) Afterward, the right room air temperature is decreased to steady values with no more changes with time. Once the temperature fluctuation becomes lower than ±0.1, the experiment is stopped, and the measurement data are extracted and processed.

The room temperature is recovered naturally with the outdoor environment until the later experiment. The next experiment is started after an approximated period of 10 hours, which assures complete recovery of the room temperature with no interfering from an earlier experiment.

4. CFD model Set-up

The three-dimensional finite volume model is developed ANSYS FLUENT software to study the heat transfer and thermal comfort conditions inside the room equipped with the RCP. Different panel arrangement patterns are suggested as shown in Fig. 3. Also dimensions for each pattern are listed in Table 1. 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. 3.

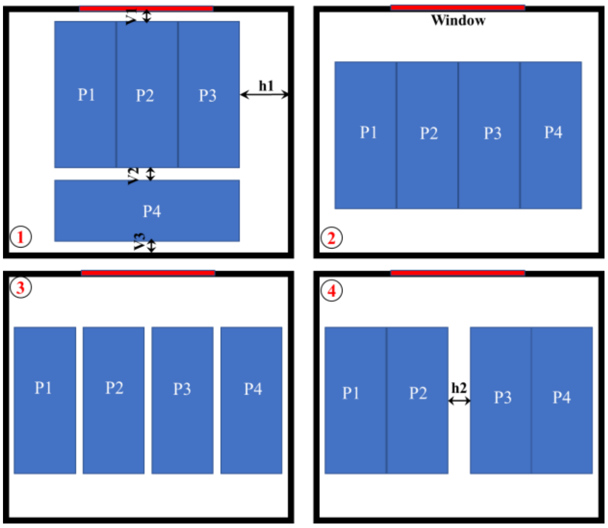

Fig. 3 Four different arrangement pattern

Table 1 Dimensions of each pattern used in this study

pattern	h1	h2	V1	V2	V3	Ap/A
1	0.52	0.06	0.11	0.11	0.18	0.6
2	0.21	0	0.46	0	0.46	0.56
3	0.08	0.08	0.46	0	0.46	0.66
4	0.08	0.14	0.46	0	0.46	0.63

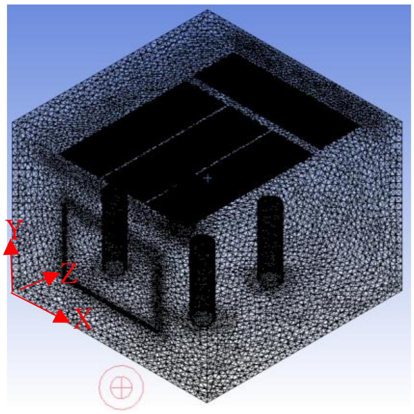

Fig.3 Grid of the simulation domain and the inflation layers

4.1. 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.

4.2. 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. The boundary conditions are summarized as shown in 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	21.3	-	0.92
Cylinder_Outer_Wall	qs=constant	-	55	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.

5. Results and discussions

The impact of different panel arrangement is examined by a numerical study. Firstly, the CFD simulation is validated with the experimental results of air temperature a long three lines at left, center and right positions. The CFD model had a good agreement with the experimental results with average error percentage of 4 %.

5.1. 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. Although, the coverage area increases from 56 % (Geom.2) to 66 % (Geom.3), the average air temperature is not changed significantly. Also, the floor temperature is not changed significantly. But in Geom.1 case where the distance between the panel and the window was 0.11 m, the average air temperature decreases by 0.5 °C compared with other arrangement. The indoor air temperature contours drawn on a vertical plane at X=1.65 m, and indoor air volume with a temperature

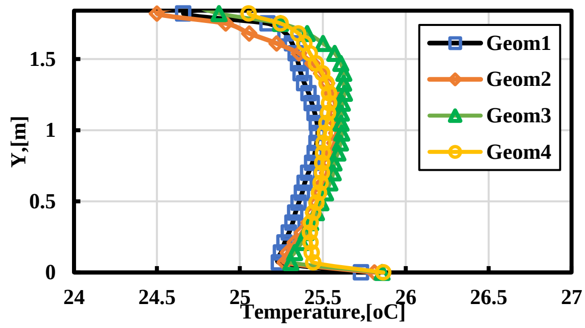

Fig.5 Vertical air temperature profile air temperature

lower than 25 °C are shown in Fig. 6.

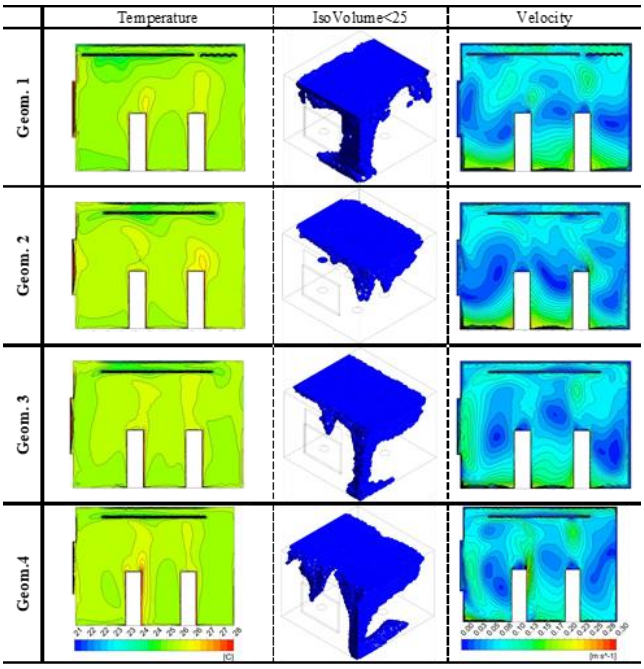

Fig. 6 Temperatures contours and Iso-Volume in each case

Figure 7 shows the average air temperature, AUST and operative temperature for different arrangement. The AUST increases from 25.9 °C to 26.1 °C for Geom.1 and Geom.2, respectively, Also, the Top decreases by 0.2 oC from 25.6 °C to 25.8 °C.

3.1.2 Heat transfer coefficients

Convection and radiation heat transfer coefficients on upper and bottom panel surfaces are calculated and summarized in Fig. 8a, b. The. Although, the differences on the bottom surface is not obvious, it is clear on the upper surface. For Geom.1, The hc and hr are 1.65 W/m^2.K and 2.13 W/m^2.K, this indicates that Geom.1 enhances the heat transfer and air movement on the upper surface.

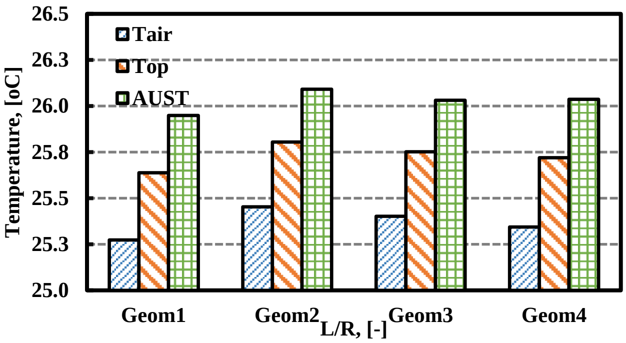

Fig. 7 Air, operative and AUST temperatures for different L/R

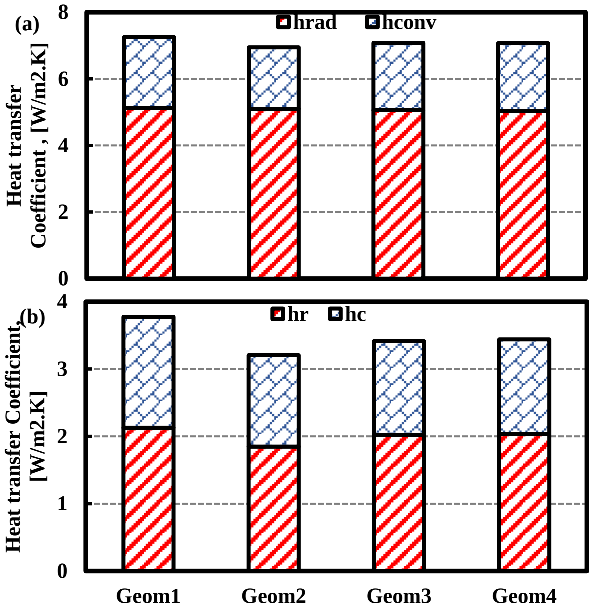

Fig8. heat transfer coefficient on a) the Bottom and b) Upper panel surfaces

3.1.3 Thermal Comfort Results

For Geom.1 the average PMV is -0.15 while the maximum is 0.4 and minimum is -0.66, also 75 % of the indoor air volume has a value of PMV lower than 0 as shown in Fig.9.

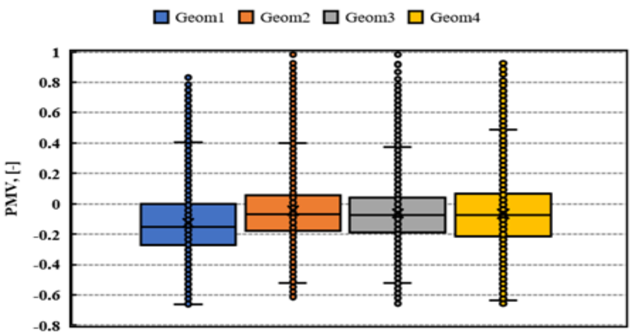

Fig. 9 Boxplot of PMV index for each geometry

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

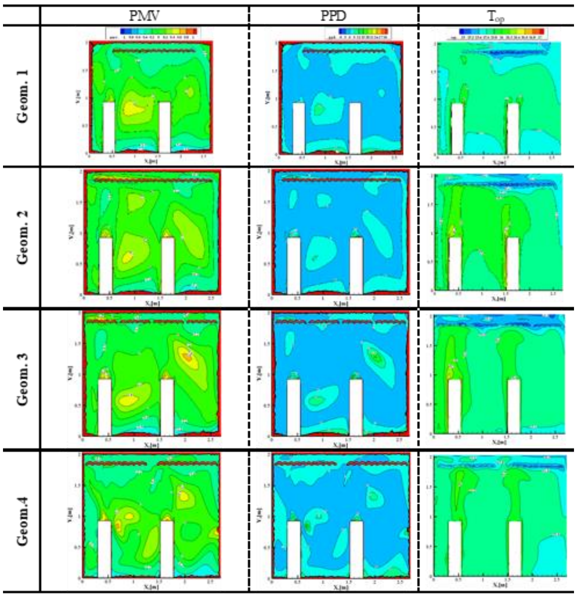

Fig. 10 PMV index, PPD index and Operative temperature.

6. Conclusions

The results are concluded as follows:

1- The CFD model had a good agreement with the experimental results with average error percentage of 4 %.

2- For Geom.1, The hc and hr are 1.65 W/m^2. K and 2.13 W/m^2.K, this indicates that Geom.1 enhances the heat transfer and air movement on the upper surface

3-The average PMV is -0.15 while the maximum is 0.4 and minimum is -0.66, also 75 % of the indoor air volume has a value of PMV lower than 0.

References

[1] Zhu X, Zhan X, Liang H, Zheng X, Qiu Y, Lin J, et al. The optimal design and operation strategy of renewable energy-CCHP coupled system applied in five building objects. Renew Energy 2020;146:2700–15.

[2] Jin W, Jing J, Jia L, Wang Z. The dynamic effect of supply water flow regulation on surface temperature changes of radiant ceiling panel for cooling operation. Sustain Cities Soc 2020;52

[3] Kong X, Xi C, Li H, Lin Z. A comparative experimental study on the performance of mixing ventilation and stratum ventilation for space heating. Build Environ 2019;157:34

[4] Rabani M, Bayera H, Nord N, Schild P, Mysen M. Performance assessment of all-air heating in an o ffi ce cubicle