Numerical Investigation on the Effect of Radiant floor panels configuration on Thermal comfort condition in Occupied zone

Muna S.Kassim1, Israa A.Abdulghafor2

1Mechanical Engineering Department, College of Engineering, Almustansiriyah University, E-mail: munskassim@uomustansiriyah.edu.iq
2Technical University (MTU) / Institute of Technology-Baghdad Electronic Technologies Department, Iraq. Israaaliabudlgaghor@gmail.com

Abstract: Radiant floor heating panels of a room with different locations is modeled by using CFD. All cases are simulate by employing k-ε equation turbulence model. After validation of code, average room air temperature, vertical air temperature difference and velocity of room air are used to predict the thermal comfort conditions in the occupied zone. Effect of indoor room air temperature and radiant floor heating panel locations on thermal conditions in occupied zone is studied. The results present that both the indoor room air temperature and radiant floor heating panels locations have effect on thermal comfort conditions and at low indoor temperature the radiant floor heating panel is insufficient to achieve thermal comfort conditions but can be reduced heating load.

Keywords: Radiant floor heating panels, Thermal comfort, PMV and PPD.

1. Introduction
The most important factors that have a direct effect on people’s health and quality of life are thermal comfort conditions. Also, thermal comfort conditions affect people’s activity [1]. Due to advantages of hydronic radiant heating and cooling panels of high energy efficiency and thermal comfort conditions, they become popular. In radiant panel about 50% or more of the total heat transfer from surface by radiation [2]. Radiant systems consume less quantity of energy and provide much more thermal comfort conditions. It is confirmed technology which achievement the requirement these are explained in related standards [3]. Radiant cooling and heating have been used for a very long time [4]. Heating system has been comprised for many years by using terminals and boiler [5]. There are many numerical studied of radiant floor heating system. Chen et al. [6] presented a numerical and experimental investigation to the performance of precast light radiant floor heating system. For analysis, they assumed as a two-dimensional heat transfer model. Simulation part studied the tube spacing and equivalent thermal resistance of the radiant panel. Simulation results are validation by experimental results. The results show that the tube space has an effect on equivalent thermal resistance. Hadavand et al. [7], studied numerically the air distribution and heat transfer in a two-dimensional enclosure when using radiant floor heating. Boussinesq approximation and control volume approach were used to solve radiation coupled with continuity, momentum and energy equations. Floor temperature, wall temperature and dimension of window and air absorptivity were simulated. The results concluded that the radiation heat transfer from the floor to air and wall in the room was about 74%. Topal et al. [8] examined experimentally and numerically the heat output of radiant panel that mounted at ceiling with different inlet-outlet temperature. The experimental results were compared with numerical results. The numerical results confirmed that the heat transfer by radiation was formed about 80% and heat output increases as
temperature increase. Jin et al.[9] investigated an original method to calculate floor temperature. Assuming the floor consists of two layers that are upper and lower, avoiding utilizing differential equation of heat transfer. The model is validate with literature experimental work, showed difference in temperature less that 2.5 °C. Previous studies focused numerically on radiant floor heating performance, air distribution and heat transfer with assumed two dimension air flows in the occupied zone. Also, the effect of radiant floor heating locations on thermal comfort conditions, air and temperature distribution in the occupied zone with different air room temperature. Since the wrong configuration can be led to discomfort in the occupied zone and cause increases in heating load. Also study the air and temperature distribution in the occupied zone and its effect on thermal comfort conditions.

2. Numerical Method

ANSYS FLUENT 14.5[10] (a commercial CFD code) is utilized to predict temperature and velocity distribution of air room with radiant floor heating. Continuity,momentum , k-€ turbulent model and Navier –Stokes equations are basic equations that use in this study. Steady state,incompressible and Newtonian flow are assumption are suggested in the present work.Calculations of indoor air flow utilize Boussinesq approximation for thermal buoyancy [11]. In momentum terms, the approximation assumes the density of air as constant and consider the action of buoyancy on air movement in the difference between weight of local air and gradient of pressure [12]. Also radiation effect takes by using radiation model.

2.1 continuity equation

The continuity equation can be written as following [13]:

\[ \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \] (1)

\( u, v \) and \( w \) are the velocity component in \( x, y \) and \( z \) direction respectively.

2.2 Navier-Stokes equation

The Navier – Stokes equations for incompressible fluid as following form in different coordinates[14]:

\[ \rho \left( \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right) = \rho g x - \frac{\partial p}{\partial x} + \mu \left[ \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right] \] (2)

\[ \rho \left( \frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right) = \rho g y - \frac{\partial p}{\partial y} + \mu \left[ \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right] \] (3)

\[ \rho \left( \frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} \right) = \rho g z - \frac{\partial p}{\partial z} + \mu \left[ \frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right] \] (4)

2.3 Energy equation

There are many ways to write the Energy Equation, one of them such as written below: [15].

\[ \rho \left[ \frac{\partial h}{\partial t} + \nabla \cdot (h \vec{V}) \right] = - \frac{\partial p}{\partial t} + \nabla \cdot (k \nabla T) + \Phi \] (5)

Where

\( h \) : specific enthalpy(kJ/kg)

\( k \) : thermal conductivity (W/m.K)

\( \Phi \) : the dissipation function represent the work done against viscous force. It can define as :

\[ \Phi = (\tau \cdot \nabla) V = \tau_{ij} \frac{\partial v_i}{\partial x_j} \] (6)
The right hand side of equation 5 is represent the pressure term.

The type of element is used in surface and volume mesh for all cases can be summarized in tables (1) and (2) respectively.

**Table (1):** Mesh surface type is used in the present study.

| Element Shape | Mesh Type | Interval Size (cm) |
|---------------|-----------|--------------------|
| Triangular    | Pave      | 1                  |

**Table 2:** Mesh volume type is used in the present study.

| Source            | Element Shape | Mesh Type | Growth rate | Max. cell size (cm) |
|-------------------|---------------|-----------|-------------|---------------------|
| Panel surface     | Tetrahedral   | T-Grid    | 1.1         | 2                   |

These mesh types is used in present study, because it’s like a free kind of mesh and suitable for any geometry without any conditions.

**2.4 Boundary conditions**

The tested room is used in the numerical study has dimensions of 200×200×300 cm in length, width and height respectively, with an insulated ceiling and walls also the dimensions of the door is 94×200 cm. The mathematical equations are solved by using ANSYS FLUENT 14.5. The objective of the numerical study in this work to analyze the velocity and temperature distribution in occupied zone when used two different arrangements of heating floor panel and different room air temperature as shown in Figure 1 with room details. The boundary conditions of the numerical study can be seen in table (1).

**Figure 1a:** Case 1 that represented the first arrangement of heating floor panel

**Figure 1b:** Case 2 that represented the second arrangement of heating floor panel

**Figure 1c:** The room details

**Table 3.** The boundary conditions are used in the numerical study

| Case Number | Interior Wall Temperature (°C) | Heating Floor Temperature(°C) |
|-------------|-------------------------------|-------------------------------|
| 1           | 12.5                          | 28                            |
| 2           | 12.5                          | 28                            |
| 1           | 14.5                          | 28                            |
| 2           | 14.5                          | 28                            |
| 1           | 16.5                          | 28                            |
| 2           | 16.5                          | 28                            |
| 1           | 18.5                          | 28                            |
| 2           | 18.5                          | 28                            |
| 1           | 20.5                          | 28                            |
| 2           | 20.5                          | 28                            |
2.5 Code validation

By the validation process the accuracy of CFD code is achieved. This validation presents comparison between predicted results from CFD code that used in the present study and the results obtained from past numerical and experimental studies. The Comparison is made by choosing one case of the present study and comparing with Seyam et al experiment and numerical study [16] with the same parameter. Figure 2 shows the temperature of the room air with height of the room.

From Figure 3 it can be seen that the same trend between two studies. Also different in value can be noticed that due to different in dimensions of tested room that was used in present study form that was used in the past study, in addition too different in operation conditions between two studies.

![Figure 2: Comparison between the present model and previous experimental and numerical data](image)

3. Thermal comfort conditions

By using the predicted mean vote (PMV) and the predicted percentage of dissatisfied (PPD) suggested by Figure 17 the thermal comfort condition can be evaluated. PMV equation can be written as following 18:

$$PMV = (0.303e^{-0.036M} + 0.028) \times [(M-W) - H - C_{res} - E_{res}]$$  \hspace{1cm} (7)

PPD is related to PMV by the following equation (19):

$$PPD = 100 - 95 \exp \left( -0.03353PMV^4 + 0.2179PMV \right)$$  \hspace{1cm} (8)

CBE tool [20] produces by Building Energy Center for determining of thermal comfort conditions according to the ASHREA standard 55-2013 is used to solve equations 7 and 8. CBE tool can be shown in Figure 2.
4. Theoretical Results

Figures 4 and 5 show the temperature contour at Y=10cm(Y is the elevation from the ground) for cases\textsuperscript{1} and 2 respectively. It’s clear that the temperature at Y=10cm increases as interior wall temperature increases due to increase room air temperature with increasing interior wall temperature. The same behavior can be seen at Y=60,110,170 cm in Figures 6, 8 and 10 for case\textsuperscript{1} and Figures 7, 9 and 11 for case\textsuperscript{2}. Figures 4 and 6 show the temperature contour at Y=10 and Y=60 cm for case\textsuperscript{1}. Its can be noticed that the temperature at Y=10cm is higher than that at Y=60cm. This is because the air temperature at Y=10cm is nearest to the heated floor. This manner can be seen in Figures 5 and 7 for case\textsuperscript{2}. Figures 6, 8 and 10 describe the temperature contour at Y=60,110 and 170 cm respectively for case\textsuperscript{1}. It’s clear that as Y increase the air temperature increases due to increase heat transfer between air layers with increase height of tested room. Same manner can be seen in Figures 7, 9 and 10 for case\textsuperscript{2}. Figures 4 and 5 present the compression between cases\textsuperscript{1} and 2 for temperature contour at Y=10cm. It can be seen that for each interior wall temperature the air temperature at Y=10cm for case\textsuperscript{2} is higher than that of case\textsuperscript{1} but the distribution of temperature for case\textsuperscript{1} is more uniform than that of case\textsuperscript{2}. Figures 6 and 7 describe the compression between cases\textsuperscript{1} and 2 for Y=60 with different interior wall temperatures. For each interior wall temperature can be noticed that the air temperature at Y=60cm for case\textsuperscript{1} is higher than that of case\textsuperscript{1}. The same behavior can be seen in Figures 8 and 9 for Y=110cm and in Figures 10 and 11 for Y=170cm.

![Figure 3. CBE tool](image)

**Figure 3.** CBE tool

![Figure 4. Temperature contours for case 1 at Y=10cm with different interior wall](image)

**Figure 4.** Temperature contours for case\textsuperscript{1} at Y=10cm with different interior wall temperatures.
Figure 5. Temperature contours for case 2 at Y=10cm with different interior wall temperatures

Figure 6. Temperature contours for case 1 at Y=60cm with different interior wall temperatures
Figure 7. Temperature contours for case 2 at Y=60cm with different interior wall temperatures.

d- Interior wall temperature 18.5°C
e- Interior wall temperature 20.5°C

Figure 8. Temperature contours for case 1 at Y=110cm with different interior wall temperatures

a- Interior wall temperature 12.5°C  b- Interior wall temperature 14.5°C  c- Interior wall temperature 16.5°C
d- Interior wall temperature 18.5°C  e- Interior wall temperature 20.5°C

Figure 9. Temperature contours for case 2 at Y=110cm with different interior wall temperatures

a- Interior wall temperature 12.5°C  b- Interior wall temperature 14.5°C  c- Interior wall temperature 16.5°C
d- Interior wall temperature 18.5°C  e- Interior wall temperature 20.5°C
Figure 10. Temperature contours for case 1 at Y=170cm with different interior wall temperatures

Figure 11. Temperature contours for case 2 at Y=170cm with different interior wall temperatures

Figures 12 and 13 show the velocity distribution at Y=10cm of cases1 and 2 respectively. It can be noticed that the velocity at Y=10cm decreases as interior wall temperature increases due to assume the air density constant at beginning solution for each wall temperature. The same behavior can be seen at Y=60,110,170 cm in Figures 14,16 and 18 for case1 and Figures 15,17 and 19 for case2. Figures 12 and 14 show the velocity distribution at Y=10 and Y=60 cm for case1. It can be concluded that the velocity at Y=60cm is less than that at Y=10cm. This is because the air temperature at Y=10cm is nearest to the heated floor so that lead to higher velocity. This manner can be seen in Figures 13 and 15 for case2. Figures 14, 16 and 18 explain the velocity contour at Y=60,110 and 170 cm respectively for case1. It’s clear that as Y increase the air velocity increases due to increasing temperature with height. Same manner can be observed in Figures 15, 17 and 19 for case 2. Figures 12 and 13 describe the compression
between cases 1 and 2 for velocity contour at Y=10cm. It can be seen that for each interior wall temperature the air velocity at Y=10cm for case 2 is less than that of case 1. Figures 14 and 15 present the compression between cases 1 and 2 for Y=60 with different interior wall temperatures. For each interior wall temperature can be noticed that the air velocity at Y=60cm for case 1 is higher than that of cases 2. The same behavior can be seen in Figures 16 and 17 for Y=110cm and in figures 18 and 19 for Y=170cm.

**Figure 12.** Velocity contours for case 1 at Y=10cm with different interior wall temperatures

**Figure 13.** Velocity contours for case 2 at Y=10cm with different interior wall temperatures.
**Figure 14.** Velocity contours for case 1 at Y=60cm with different interior wall temperatures

- **a:** Interior wall temperature 12.5°C
- **b:** Interior wall temperature 14.5°C
- **c:** Interior wall temperature 16.5°C
- **d:** Interior wall temperature 18.5°C
- **e:** Interior wall temperature 20.5°C

**Figure 15.** Velocity contour for case 2 at Y=60cm with different interior wall temperatures

- **a:** Interior wall temperature 12.5°C
- **b:** Interior wall temperature 14.5°C
- **c:** Interior wall temperature 16.5°C
- **d:** Interior wall temperature 18.5°C
- **e:** Interior wall temperature 20.5°C

**Figure 16.** Velocity contours for case 1 at Y=110cm with different interior wall temperatures

- **a:** Interior wall temperature 12.5°C
- **b:** Interior wall temperature 14.5°C
- **c:** Interior wall temperature 16.5°C
- **d:** Interior wall temperature 18.5°C
- **e:** Interior wall temperature 20.5°C
Figure 17. Velocity contours for case 2 at Y=110cm with different interior wall temperatures

Figure 18. Velocity contours for case 1 at Y=170cm with different interior wall temperatures

Figure 19. Velocity contours for case 2 at Y=170cm with different interior wall temperatures
5. Thermal comfort conditions

Figures 20 and 21 present the thermal comfort condition on comfort chart, in addition to calculate PMV and PPD via CBE tool for cases 1 and 2 respectively. It’s clear that as interior wall temperature increases the comfort conditions is achieved due to decrease PPD and approach PMV from neutral state for each case. Also can be noticed that the thermal comfort conditions is the best in cases 1.

![Image of thermal comfort conditions charts]

**Figure 20.** Thermal comfort conditions for case 1 with different interior wall temperatures

a- Interior wall temperature 12.5°C  
b- Interior wall temperature 14.5°C  
c- Interior wall temperature 16.5°C  
d- Interior wall temperature 18.5°C  
e- Interior wall temperature 20.5°C
6. Conclusion

Numerical study to investigate the effect of radiant heating floor panel configuration on thermal comfort conditions for different air room temperature has been studied. The data created by models that represent the complexity of the flow patterns in the heated space is analyzed by CFD. The k-ε equation is used in this investigation. Different configurations of radiant heating floor panel and different air room temperature effect on distribution of air velocity and temperature. For the different radiant heating panel configuration and for each air room temperature, the case1 can be achieved thermal comfort conditions more than that of case2. Also, for different air room temperature and for each radiant heating panel configuration, it can be concluded that as air room temperature increases the thermal comfort conditions are achieved.

7. Nomenclature

Cres: Respiratory convective heat exchange (W/m²)
E: Evaporative heat exchange at the skin (W/m²)
Ec: Evaporative heat exchange at skin (W/m²)
Eres: Respiratory evaporative heat exchange (W/m²)
Esw: Evaporative heat loss from evaporation of sweat (W/m²)
H: Dry heat loss (W/m²)
M: Metabolic rate (W/m²)
T: Temperature (°C)
W: Effective mechanical power (W/m²)

Abbreviations
ASHRAE: American Society For Heating, Refrigeration And Air Conditioning Engineers
CFD: Computational Fluid Dynamic
PMV: Predicated Mean Vote
PPD: Percentage People Dissatisfied

8. References

[1] Olesen, B., 2008, Radiant floor cooling systems, ASHRAE Journal, vol. 50, no. 9, pp. 16–22
[2] ASHRAE, 2008, ASHRAE Handbook - Inc, Atlanta, GA.
[3] ANSI/ASHRAE, 2013, Standard 55 Inc, Atlanta, GA.
[4] Bean, R.; Olesen, B.W.; Kim, W.W. 2010 History of Radiant Heating & Cooling Systems, Part 2. ASHRAE J. 52, 50–55
[5] Giovanni Ferrarini 1, ID , Stefano Fortuna 1, Alessandro Bortolin, Gianluca Cadelano, Paolo Bison, Fabio Peron and Piercarlo Romagnoni, 2018 Numerical Model and Experimental Analysis of the Thermal Behavior of Electric Radiant Heating Panels, *Appl. Sci.* 8, 206.

[6] Guoqing Yu,*, Wei Guo, Hengtao Chen, 2017 Thermal performance of precast light floor radiant panel, *Procedia Engineering* 205 198–204

[7] H.Khorasanzadeh,G.A.Sheikhzadeh,AA.Azematii,B.Shirkavand Hadavand. . 2014, Numerical study of air flow and heat transfer in a two –dimensional enclosure with floor heating. *Energy and Buildings* 78, 98-104

[8] Eda Ergur, Serhat Unver and Tamer Calisir, 2018 Experimental and numerical investigation of ceiling mounted radiant panels heat output for different inlet-outlet water temperatures *International Journal of Scientific Research in Information Systems and Engineering*, volume 4, issue 1.

[9] Jin, X.; Zhang, X.; Luo, Y. 2010 A calculation method for the floor surface temperature in radiant floor system. *Energy Build*, 42, 1753–1758.

[10] ANSYS FLUENT 14.5 (2010) program.

[11] D. etherize, M. Sandberg, 1996 *Building ventilation, theory and measurement*, England: John Wiley and Sons Ltd.

[12] Ramy H. Mohammed, 2014 Numerical Investigation of Displacement Ventilation Effectiveness, *International Journal of Environmental and Ecological Engineering* vol:8, No:2.

[13] A. Salih 2011 *Conservation Equations of Fluid Dynamics* Indian Institute of Space Science and Technology, Thiruvananthapuram.

[14] Sylvio R. Bistafa 2017 *On the development of the Navier-Stokes equation by Navier* University of de São Paulo, São Paulo, SP, Brazil.

[15] R.C.HIBBELEER,2015, book of Fluid Mechanics

[16] Shaimaa Seyam,Ahmed Huzayyin,hesham El-Batsh and Sameh Nada,2014 Experimental and numerical investigation of the radiant panel heating system using scale room,*Energy and Buildings* 82, 130-141.

[17] Fanger, P.O., 1984. Moderate thermal environments determination of the PMV and PPD indices and specification of the conditions for thermal comfort. *ISO 7730.*

[18] Fuzhong Nian and Ke Wang 2017 Study on indoor environmental comfort based on improved PMV index *Computational Intelligence & Communication Technology (CICT)*, 2017 3rd International Conference on

[19] Q. Chen, W. Xu, 1998 A zero-equation turbulence model for indoor airflow simulation *Energy and Buildings* 28, 137-144

[20] F.Bauann 2012 *Thermal Comfort Research at the Center for the Built Environment Center for the Built Environment (CBE)* the University of California.