Kathmandu University Journal of Science, Engineering and Technology, Vol. 15, No. 2, August 2021

Kathmandu University
Journal of Science, Engineering and Technology

CFD analysis of temperature distribution of different piping arrangement used in radiant floor heating system

Ananta Aacharya *, Shuvas Khanal, Bishal Humagain, Sangeet Kattel, and Bivek Baral

Department of Mechanical Engineering, Kathmandu University Dhulikhel, Nepal.

Abstract

The rate of energy consumption on the household level for heating and cooling is increasing annually. Meeting future heating and cooling energy demand by alternative and promising technology like floor heating systems (FHS) should be the primary concern for engineers and designers. The floor heating system (FHS) maintains the desired indoor temperature with low-temperature fluid flowing inside the pipe than conventional heating systems. FHS has lower investment cost, lower energy consumption, better thermal comfort, and desired temperature up to human height. This paper aims to simulate and compare the temperature distribution for three different piping arrangements like double-wall serpentine, serpentine, and modulated spiral. The geometry was created in Solidworks. The system was modelled using sets of boundary conditions. The simulation results show that the spiral layout has a temperature distribution of 320 K at an outlet and has a temperature gradient of about 13°C between the inlet and outlet. This arrangement generates and heats the air uniformly in the room and provides better thermal comfort. Even serpentine has a small temperature gradient, and this system creates a periodic cycle of air movement in the room, which is the worst case in terms of thermal comfort. Doubled wall serpentine has a temperature gradient of 20°C, and this arrangement is the worst among the three layouts. The study concluded that spiral arrangement has a uniform temperature distribution among all the setup, heats the air uniformly, and provides better thermal comfort.

Keywords: CFD; Floor heating system; Temperature distribution; Simulation

---

1. Introduction

People use electrical appliances like air conditioners and heaters for thermal comfort with improved living standards and economic growth. The heating and cooling sectors use 10% of energy consumption on Nepalese households [1]. This percentage is increasing annually. Thus to meet the future heating and cooling energy demand in the building sector, a solar-assisted FHS might be an appropriate technology. The FHS is a form of heating system that maintains desired indoor temperature through heat transfer between radiant surface and room by conduction, convection, and radiation. It consists of a piping arrangement embedded through which hot water flows and heat transfer occurs. These heating systems apply in residential and commercial buildings such as hotels, banks, educational institutions, and airplane hangar. The FHS system has a low investment cost, low energy consumption, better thermal comfort, and desired temperature up to human height. Researchers conducted many studies on FHS regarding thermal performance, heat transfer phenomenon, pressure drop, and friction loss in pipe surfaces and system arrangement.

Better thermal performance, maximum energy efficiency, uniform temperature distribution for thermal comfort, and energy conservation are the designer's primary design considerations and ultimately for consumers. The performance and optimization depend on several parameters such as piping layout, pipe spacing, initial temperature, and mass flow rate of the fluid. It is common to use two types of piping arrangements in the FHS, namely serpentine and spiral. These arrangements have different temperature distribution on the floor, directly impacting the system's thermal performance.

The layout selection depends on the room's heating load, the wall's thermal properties, number of windows and door. The single wall serpentine is used for a comparatively small room to distribute more heat energy toward the high heat loss wall surface. It is expected that the room has more than one wall exposed to the environment. The heat loss rate from these walls is maximum and in such case, doubled wall serpentine layout is preferable. The double-wall serpentine configuration delivers more heat energy toward these surface to equalize the heat loss from the wall surface. The spiral arrangement could be the possible option for the large room with uniform heat loss from all the surface. This configuration has the supply and returns portions of the loop immediately adjacent to one another.

Energy conservation and energy efficiency are other parameters that need to address while designing the heating system. For low-temperature heating systems, the aim should be an efficient use of energy and sacrifice thermal comfort for energy conservation. Heating systems need to provide uniform temperature distribution that ensures better thermal comfort. The different simulation approaches help to optimize the system's performance. The simulation of FHS has two significant parts, namely the floor of the room and internal space. The foundation includes pipes, hot fluid, space between lines and floor generally, wood or concrete. Similarly, the inner area consists of the room, occupants, walls, ceiling, windows, door, and air within the zone. The floor is a thermal energy source; changing floor properties directly impacts room temperature.

This paper's main objective is to conduct the simulation of the temperature distribution of different piping arrangement used in the radiant floor heating system. In this paper, a simulation of the FHS system was limited to thermal analysis of the zone pipe flow, assuming that zone temperature will be uniform when the pipe has

---

*Corresponding author. Email: aacharya.ananta123@gmail.com uniform temperature distribution. The temperature distribution in the pipe section for three different piping arrangements was analyzed and compared. The pipe lining with almost constant temperature distribution provides better thermal comfort in the zone. The study of the impact of varying the flow rate on the temperature distribution and the outlet temperature was the second part of the study.

1.1. Literature review

The numerical simulation for analyzing the performance of the heating system has become a popular approach. It helps to solve complicated engineering equations and problems more efficiently. It encourages researchers and engineers to solve fluid and heat transfer problems in the FHS. The numbers of research in the floor heating system have been conducted in terms of thermal perspective, thermal comfort, the temperature distribution in floor & room by researchers worldwide.

Flavio A. Damasceno et al. [4] analyzed the CFD model that predicts the surface temperature in the heating of a farrowing house and found that an alternative heating system was efficient in maintaining the floor temperature required for thermal comfort. They created the geometry in ANSYS ICEM with a tetrahedral mesh. The results showed that measured and predicted value has normalized mean square error of 0.002 and 0.0005 for conventional and alternative treatment and proved that CFD is an efficient design analysis tool.

Yinghui et al. [2] analyzed indoor airflow's thermal performance in the heated room via the radiator, air conditioner, and the FHS. A numerical simulation studied the temperature, pressure, and velocity field for a residential building of a 20m^2 area with an indoor temperature of 18°C and an outdoor temperature of -9°C The simulated room was located on the building's middle floor, assuming zero radiation effect, incompressible airflow in the steady-state turbulent flow. The numerical solution with hexahedral structured grids showed that FHS provides a more uniform temperature field and lower airflow velocity. Instead of numerical simulation for the room, two-dimensional modeling was done for floor heating and concentrated heating for an enclosure. The results also show that floor heating generates a more uniform temperature than conventional heating [3].

Xuejing Zhang et al. [5] proposed a model to calculate surface temperature and heat output assuming air as incompressible medium, heat transfer process in steady-state. The three-dimensional numerical model built using fluent for solving continuity, momentum equations, and energy equations using RNG, k-ε model for turbulence, and discrete ordinate (DO) for radiation model. The result showed that radiant floor heating gives the temperature of 27.8°C, which is the required temperature for thermal comfort.

To analyze FHS in a dome shape and cubic structure, T. Khademinejad et al. [6] simulated a dome-shaped house by solving a governing equation. They used k-ω and DO for the turbulence model and the radiation model, respectively. The three dimensional model of the dome and cubic shape showed that the heat transfer from the floor for the dome shape is 6.5% more than the cubic form. The floor heat transfer area also decreased by 23% with uniform temperature distribution in the dome-shaped room.

Sarika kumara Mishra et al. [7] modelled and simulated the underfloor heating for two different pipe configurations. The floor with the radiant heating system solved numerically using RNG k-ε in ANSYS's academic release. The simulation results showed that the spiral loop generates uniform temperature distribution and is an efficient configuration.

Some parameters, like velocity at the inlet, inlet temperatures, the tubing pattern, etc., were studied by creating the unstructured triangular mesh for a solid model with progressive increment in cell number [8]. The results showed a consistent average temperature for spiral configuration, and grid-independent study had no impact on the floor's surface temperature.

Liang Wang et al. [9] studied an application of solar air collectors and the air supply heating system in winter in a room of 20 m^2 area. The results were monitored for floor air supply system in fluent for 500 times showed that the change of outlet temperature was 328K, meeting the standard for winter.

Miroslaw Zukowski et al. [10] conducted a numerical analysis of air underfloor heating systems and built a model to predict velocity and temperature distribution. A two-dimensional model of the room with a height of 2.5m and a width of 4m created in CFD, and governing equations were solved using the finite element method. The model met the convergence criteria by changing the mesh type, and the results have a good agreement between simulated and experimented data. The results showed a uniform profile of the temperature in the middle from floor to ceiling. The location of the outlet vent has no impact on temperature distribution.

2. Methods

2.1. System geometry and model description

In this study, FHS was simulated for the pipe section using Computational Fluid Dynamic and compared the results in terms of temperature distribution for different piping arrangements. The floor model has an area of 14.07 m^2 for each case. The pipe shaping and the piping arrangement significantly impacts heat output, thermal performance, and temperature distribution on the floor. The model included the three different piping arrangements embedded in the floor area, having a pipe diameter of 20 mm and a spacing of 200 mm. Three piping layouts, namely serpentine, doubled wall serpentine, and spiral, are shown in Fig. 1.

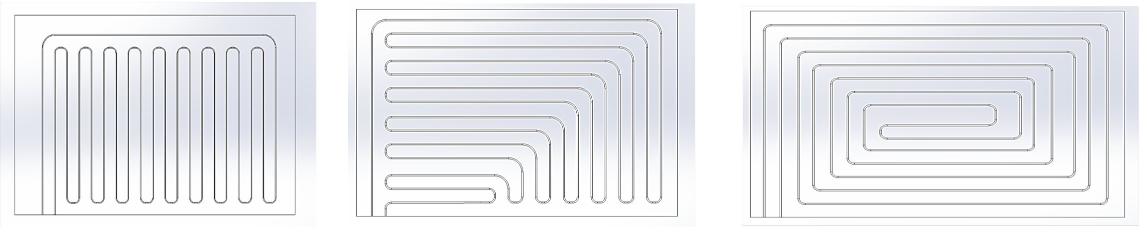
A single room of a residential building was considered an enclosure with a spiral piping arrangement embedded within the floor shown in Fig. 2. The model has a dimension of 4.57 m× 3.80 m × 2.80 m does not contain doors, windows, and any other components. The temperature distribution and airflow in the room is a complex phenomenon. Thus, the assumptions were:

- Flow in the pipe is considered incompressible.
- Steady-state condition assumption.
- The room is an enclosure that does not have doors, windows, and occupants.

Thermal analysis of the FHS has two significant sections- floor and internal space. The foundation contains heating pipes, concrete or wood, floor covering, while the interior area includes air, occupant, wall, ceiling, doors, and windows. In this study, the simulation was limited to pipe layout. The simulation study has two parts: pipe simulation for temperature distribution and pipe simulation for the same configuration with varying flow rates. The first simulation's philosophy is that uniform pipe temperature heated up the air above it at a uniform rate, thus providing uniform temperature inside the room. The second simulation shows the impact of changing the mass flow rate on outlet temperature and temperature distribution on the piping layouts.

2.2. Numerical setup

The air properties inside the room are directly dependent on the floor's thermal performance. The first part of the study includes the simulation of temperature distribution. In the second part, the simulations run for the same configuration with varying the mass flow rate to study the effect of flowrate on the temperature distribution. In this study, the simulation runs for three different piping layouts doubled wall serpentine, serpentine, and spiral having



(a) Serpentine (b) Doubled wall serpentine (c) Spiral

Figure 1: Three different piping layout of floor heating system

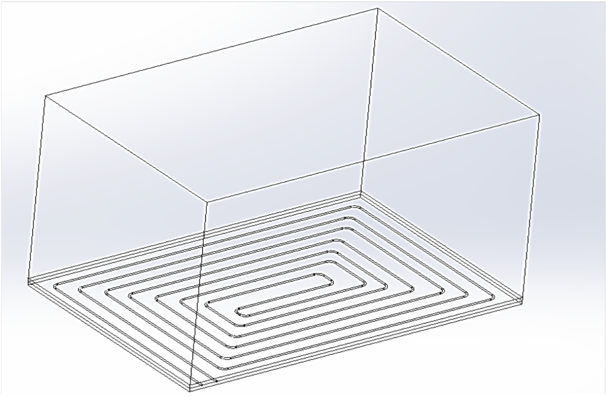


Figure 2: Room geometry spiral arrangement embedded within the floor.

Table 1: Boundary conditions.

Case	Boundary Condition	Location	Value
A	Mass flow inlet Inlet temperature	Inlet	32.7 g/s 333 K
	Outflow	Outlet	Mass flow weighting: 1
	Pipe Wall Free stream temperature	Wall	40 W/m^2 K 295 K
B*	Mass flow inlet	Inlet	65.4 g/s

*B: For case B, all boundary conditions are the same except the inlet mass flow rate.

20mm pipe diameter and 200mm spacing with the same flow rate for all cases.

The numerical model was set in ANSYS Fluent 15.0, which uses FVM to solve the differential equation numerically. A model has a pipe configuration embedded within the floor. The governing equations were mass, energy, and momentum, solved for steady-state conditions using the first-order upwind for momentum equation and the second-order upwind for energy equation. The model had a structured numerical grid of 863856 cells, 989835 cells, and 1015630 cells for doubled wall serpentine, serpentine, and spiral layout. The model with a grid of doubled wall serpentine (a), serpentine (b), and spiral (c) layouts is depicted in Fig. 3. The turbulence model was RNG k-ε with enhanced wall treatment. The convergence criteria for the solution was 10^{-4}.

2.3. Boundary condition

This study focused on the steady-state simulations for different piping layout embedded within the floor. The inlet water temperature and flow rate were assumed and doubled for the next case. The second case has the same temperature condition with a doubled mass flow rate to study the system's temperature distribution. The inlet has mass flow inlet and inlet temperature; the outlet has outflow condition. The wall condition has the heat transfer coefficient with the free stream temperature as the boundary condition. Table 1 shows the numeric value used for boundary conditions.

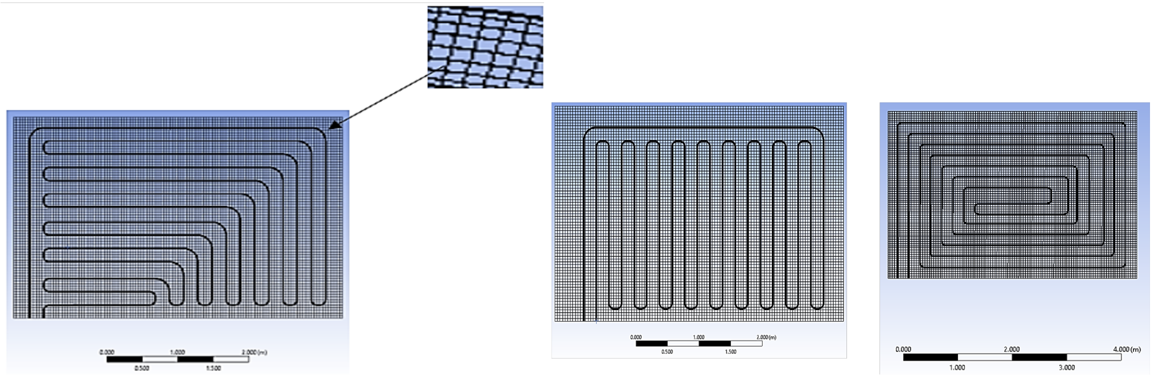
3. Results and discussion

3.1. Temperature distribution analysis of doubled wall serpentine

In the FHS analysis, the pipe temperature and distribution are the crucial parameters that determine the floor surface temperature and room temperature distribution. Fig. 4 shows the temperature distribution of fluid in the doubled wall serpentine arrangement. The result shows a small region with a temperature above 314 K. Most of the area has a temperature below 314 K. This large temperature gradient between the inlet and the outlet region creates a periodic heating cycle in the room. Using this arrangement gives a sensation of intermittent heating to the occupants. There is a considerable effect of changing mass flowrate on the system. In the second case, the fluid flow rate was adjusted to study the impact of varying mass flow rate on the temperature distribution. Increasing the mass flow rate generates most regions with a temperature between 310 K and 315 K, having a temperature gradient of only 5 °C. This result means that the water leaves the circuit at a higher temperature, which carries the potential energy. Lastly, the outward inlet has a fair temperature distribution than the inward inlet and suggested avoiding this arrangement in most cases.

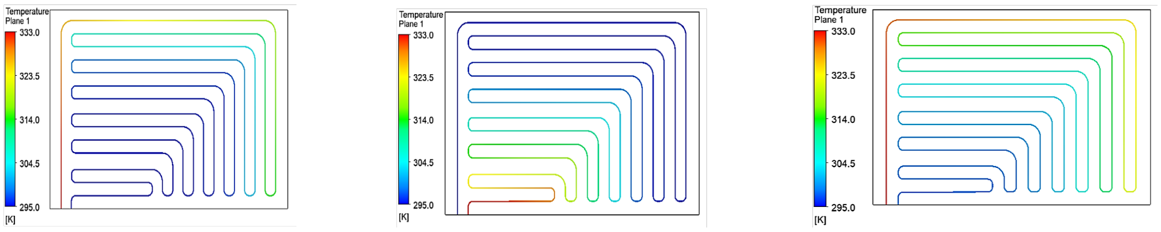
3.2. Temperature distribution analysis of the serpentine configuration

Fig. 5 shows the temperature distribution for the serpentine configuration. The outlet temperature is above 314K for all the cases, which indicates a small temperature gradient between the inlet and the outlet. The simulation result in Fig. 5 (b) shows that the inlet should place toward the heating loop. Even serpentine arrangements have a small temperature gradient; this system heats the air in one side much faster and thus creating a periodic cycle of air movement. Fig. 5 (c) shows the simulated results for case B boundary condition. There is a significant effect of varying mass flowrate on the outlet temperature of the fluid. Increasing the mass flow rate increases the fluid temperature at the outlet and thus reduces the temperature gradient. The liquid temperature leaving the circuit is around 325K with a temperature gradient of only $8^\circ C$. The system with an inward inlet is suggested as this case will create a more uniform temperature distribution than in Fig. 5 (a). In this case, water leaves at a higher temperature reducing the possibility of using the potential energy of outflow liquid. One option of the increasing temperature gradient is lowering the tube spacing so that the liquid spends more time in the heating circuit and, finally, increased the heating circuit's efficiency. However, the lower flow



(a) doubled wall serpentine (b) serpentine (c) spiral

Figure 3: Mesh generation for piping layout.



(a) Inlet outward, case A boundary condition (b) Inlet inward, case A boundary condition (c) Outward inlet, case B boundary condition

Figure 4: Temperature distribution for doubled wall serpentine arrangement.

rate system with moderate spacing is the energy-efficient system.

3.3. Temperature distribution analysis of spiral layout

Fig. 6(b) shows a somehow more uniform temperature distribution generated by an inward inlet spiral loop with minimal thermal gradient. These arrangement types generate and heat the air in the room more uniformly and generate a low air velocity filled in the room and better thermal comfort. The optimized spiral layout in Fig. 6 has a temperature of 320K at the outlet and has a temperature gradient of about 13°C between the inlet and outlet. Fig. 6 (b) shows the change in the temperature distribution by changing the inlet position. The water is likely to leave at a higher temperature. This layout generates a more uniform temperature distribution. Fig. 6(c) shows the effect of increasing the mass flow rate on the system's temperature distribution. The increase in mass flow rate decreases the temperature gradient. The outlet temperature of the fluid is around 325K having a temperature gradient of 8°C. There is a more uniform temperature distribution than the previous case. The outgoing fluid has the potential thermal energy that can be utilized. The potential outgoing energy can be used by decreasing the tube spacing. The reduced space increases the time spent on the fluid in the heating circuit. However, a lower flowrate system with moderate spacing suggested by the manufacturer is the basis for designing the system.

3.4. Comparison with other studies

Most of the numerical methods applied for this study are nearly similar to that of several other studies (Table 2). However, the study of changes in the mass flow rate on the temperature distribution and comparing different piping arrangements are different from others. The set of boundary conditions, inlet temperature, mass flow rate, tube spacing, and pipe diameter are different. The study performed a simulation for the same tube spacing and pipe diameter for the different piping arrangements and compared them.

This study's significant difference was the temperature distribution for all the cases were simulated in the pipe. The room was out of consideration for simulation, and the change in temperature field, velocity, and pressure field in the room space was out of scope. This analysis's basic philosophy is that the uniform temperature distribution in the pipe flow will heat the air uniformly. The system with consistent temperature will meet the thermal comfort requirements quickly.

Among all cases, spiral layout with inward inlet and case B boundary condition generates the uniform temperature distribution with a minimal temperature gradient of 8°C. This layout heats the air in the room uniformly and provides better thermal comfort. The worst-case was the doubled wall serpentine configuration. Most of the region has a temperature below 314 K that creates a periodic heating cycle in the room and gives the sense of intermittent heating to the occupants. The researches on the heating circuit configuration were conducted to investigate the temperature distribution. The study is similar to them, but the names of the arrangements are different.

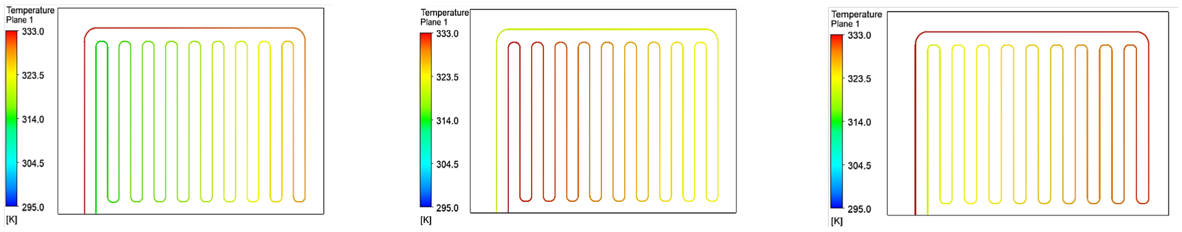
4. Conclusion

In this study, the CFD simulation for temperature distribution on the pipe surface was studied for the different piping layout. The temperature distribution for different piping configuration was analyzed and compared. The spiral arrangement has a temperature distribution of 320 K at the outlet. The same system has a temperature gradient of about 13°C between the inlet and the outlet.

This arrangement generates and heats the air in the room uniformly and provide better thermal comfort. Even serpentine has a small temperature gradient; this system creates a periodic cycle of air movement in the room, which is the worst case in thermal comfort. Doubled wall serpentine has a temperature gradient of 20°C. Most of the region has a temperature between 310 K and 314 K. This configuration is the worst among the three layouts. The radiant heating system with this arrangement creates periodic heat-

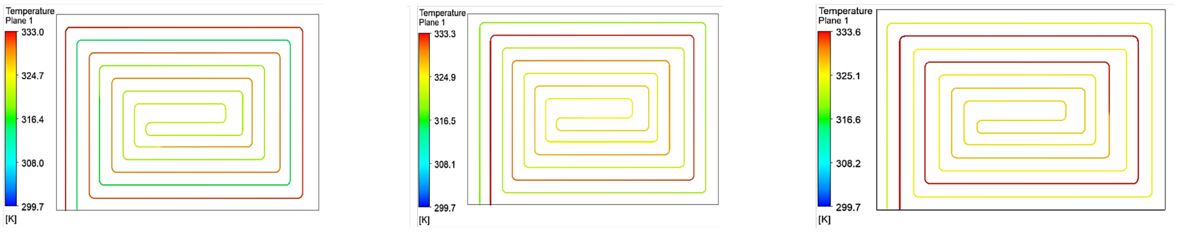
Table 2: Comparison of various studies with their system outcome.

Authors	System Features				Boundary Conditions	CFD Methodology	Results
	Country	Building	Area	System			
Our Study	Nepal	Simple Floor	$17m^2$	Radiant heating system	Inlet temperature: 333K Mass flow inlet: 32.7g/s Outlet Condition: Outflow Pipe wall: Convective heat transfer coefficient: $40W/m^2K$ Free Stream temperature: 295K Case B: Mass flow rate doubled	CFD code: FLUENT Pressure-velocity coupling: SIMPLE Scheme: Second order Upwind Scheme Turbulence model: RNG k-ϵ Mesh: Finest structured mesh	The spiral loop generates a more uniform temperature distribution with a minimal temperature gradient of 13°C. This layout heats the air in the room more uniformly and provides better thermal comfort. Increasing mass flow rate creates a more uniform temperature distribution. However, the water leaves at a higher temperature. The temperature gradient is just 8°C.
Mishra et al. [7]	Nepal	Restaurant	$89.76m^2$	Solar-Floor heating system	Mass flow inlet Solid Domain Inlet temperature: 311K No radiation effect	CFD code: FLUENT Pressure-velocity coupling: SIMPLE Turbulence model: RNG k-ϵ Scheme: Second order upwind scheme Mesh: Unstructured mesh	The counter flow spiral loop creates a uniform temperature distribution with a gradient of 5°C. The tube spacing should not be increase drastically as it causes thermal fatigue in the floor, as a result of which cracks developed
Wang et al. [2]	China	Residential	$20m^2$	Floor Heating System	Natural Convection Adiabatic Wall Heat flux: 0Surface temperature: 20°C Floor thickness: 0.012m	CFD code: FLUENT Pressure-velocity coupling: SIMPLE Turbulence model: k-ϵ Grid independence study Mesh: Hexahedral structured grid using GAMBIT	The interior temperature of the radiant heating model is 21.42°C. The air temperature at human feet is 24.13°C. The temperature is uniform in the longitudinal direction and has a better thermal comfort
Khorasanizadeh et al. [3]	Iran	Simple enclosure	$9m^2$	Floor Heating System	Indoor design temperature: 20°C Radiation Effect Floor temperature: 22°C Ceiling temperature: 23°C Window temperature: 11°C Heat transfer coefficient: Floor: $0.582W/m^2K$; Ceiling:0.2682 W/m^2K; Wall: 0.4314 W/m^2K; Window: 3.688 W/m^2K	Continuity, momentum, turbulence, and energy equation solved in 2D Cartesian coordinate Pressure-velocity coupling: SIMPLE Turbulence model: Modified k-ϵ Discrete Ordinates Radiation model Mesh: Structured grid	The vertical temperature gradient in the floor heating system is closed to the ideal thermal comfort distribution.The temperature distribution in the floor heating system with the spiral arrangement is more uniform, and the heat flux of the system is 156.48W
Izadi et al. [11]	Iran	Simple room	$12m^2$	Floor Heating System Parallel model Reciprocating model	Mass flow inlet: 1.3-1.5lit/min Outflow Inlet temperature: 35-55°C	CFD code: Fluent Pressure-velocity coupling: SIMPLE Scheme: Second order Upwind Mesh: Using GAMBIT Wall: Triangular mesh Pipe: Quadrilateral mesh Room space: Unstructured Four Wedge	Parallel mode: The room temperature is uneven. The warmest place is farther from the window, and the lowest temperature nearest the windowReciprocating model: The arrangement gives a more uniform temperature, and the room achieved thermal comfort condition quickly.



(a) Inlet outward, case A boundary condition (b) Inlet inward, case A boundary condition (c) Outward inlet, case B boundary condition

Figure 5: Temperature distribution for a serpentine arrangement.



(a) Inlet outward, case A boundary condition (b) Inlet inward,case A boundary condition (c) Inward inlet,case B boundary condition

Figure 6: Temperature distribution for spiral arrangement.

ing where the occupant feels the intermittent heating. The position of the inlet has a significant impact on the temperature distribution. It is better to have an inward inlet for the spiral and the serpentine configuration as the distribution is more uniform for this case. However, the temperature gradient of the system gets reduced. The gradient can be increased by lowering the tube spacing. The fluid will spend a long time passing through the circuit, but the low flow rate system is energy-efficient. Among all the cases, the spiral configuration with moderate flow and spacing inward inlet system creates a uniform temperature distribution and is an energy-efficient system.

References

[1] Shrestha S M, *Energy Efficient Building for Nepalese Market.* Master Thesis, Institute of Building, Energy & Material Technology, The Artic University, Norway, 2017.

[2] Wang Y, Zhang X, Ji J, Tian Z, & Li Y, Numerical simulation of thermal performance of indoor airflow in heating room, in *Proceedings of 10th International Conference on Applied Energy (ICAE2018)*, Hong Kong, China, 22-25 August 2018.

[3] Khorasanizadeh H, Sheikhzadeh G A, Azemati A A & Hadavand B S, Numerical study of air flow and heat transfer in a two-dimensional enclosure with floor heating, *Energy and Buildings*,78 (2014) 98–104.

[4] Damasceno F A, Oliveira C E A, Saraz J A O, Schiassi L & de Oliveira J L, Validation of a Heating System in the Farrowing House using a CFD Approach, *Engineering Agrícola*, 38(4) (2018) 471–477.

[5] Zheng X, Han Y, Zhang H, Wandong Z & Kong D, Numerical study on impact of non-heating surface temperature on the heat output of radiant floor heating system, *Energy and Buildings*, 155 (2017) 198–206.

[6] Khademinejad T, Rahimzadeh S D, Sardari P T & Sarkardeh H. *CFD Modelling of Floor Heating System in Dome Shape Rooms according to the Thermal Comfort Condition*, In: M. editors. Maher A.R. Sadiq Al-Baghdadi, Najaf, Computational Fluid Dynamics Application in Green Design, Iraq (2014), ISBN 1494875756.

[7] Mishra S K & Bajracharya T R, Design and Numerical Analysis of Solar Underfloor Heating System: A Case Study of Resort in Nagarkot, Nepal, in *Proceedings of IOE Graduate Conference*, Kathmandu, 2017.

[8] Valipour M S & Ghadi A Z, Numerical investigation of fluid flow and heat transfer around a solid circular cylinder utilizing nanofluid, *International Communication in Heat and Mass Transfer*, 38 (2011), 1296-1304.

[9] Wang L, Man Y, Shi S, & Wang Z, Application of Solar Air Collector and Floor Air Supply Heating System in Winter, in *Proceedings of 10th International Symposium on Heating, Ventilation and Air Conditioning, ISHVAC2017*, Jinan China, 19-22 October 2017.

[10] Zukowski M, A Numerical Analysis of Heat and Mass Transfer in a room with the air-underfloor heating, in *Proceedings of Eighth International IBPSA Conference*, Eindhoven, Netherlands, 11-14 August 2003.

[11] Izadi M J & Makaremi K, CFD simulation of temperature distribution in a room with various under floor heating system models, in *Proceedings of ASME 2009 Fluids Engineering Division Summer Meeting,* Vail, Colorado USA, 2-6 August 2009.