

## IMPACT OF ROOM HEIGHT ON HEAT TRANSFER EFFICIENCY IN VARIOUS HEATING SYSTEMS: A NUMERICAL ANALYSIS

Abdolali Sabeti Monfared<sup>1</sup>, Ahmad Moghaddasi<sup>2\*</sup>, Mojtaba Pourgholamali<sup>3</sup>

<sup>1</sup> Department of Engineering, Omran Abshar Espadana Co., Tehran, Iran

<sup>2</sup> School of Design and the Built Environment, Faculty of Arts and Design, University of Canberra, Canberra, ACT, Australia

<sup>3</sup> Faculty of Civil Engineering, Islamic Azad University, Najafabad, Iran

\*Corresponding author's email: Ahmad.moghaddasi@canberra.edu.au

### Abstract

**The purpose of the study.** This study investigates the impact of varying room heights on the heat transfer efficiency of heating systems, specifically comparing conventional radiator-based heating with underfloor heating. **Methods.** Using Computational Fluid Dynamics (CFD) simulations, we analysed temperature distributions at three ceiling heights (2.7 m, 3 m, and 3.3 m). The numerical model was used to examine airflow behaviour, thermal stratification, and temperature variation within the occupied space under different heating conditions. **Results.** The results indicate that lower room heights tend to reduce vertical temperature stratification, thereby increasing the overall perceived temperature, while taller rooms exhibit greater temperature gradients between the floor and ceiling. The simulations also show that the location and mode of heat delivery have a substantial influence on indoor temperature distribution. Furthermore, underfloor heating systems demonstrated superior thermal uniformity and energy efficiency compared to radiator-based systems by distributing heat more evenly across the lower occupied zone. **Conclusions.** These findings contribute to optimizing energy efficiency and occupant comfort in building design by tailoring heating strategies to specific room dimensions. The study also highlights the importance of considering room geometry alongside heating-system selection when evaluating indoor thermal performance and developing energy-conscious design solutions for enclosed indoor spaces.

**Keywords:** floor heating, heat transfer, natural displacement, computational fluid dynamics.

### Introduction

Energy consumption has increased significantly over the past few decades, primarily driven by industrial growth and rapid urbanisation, which have led to the proliferation of buildings and infrastructure. In Iran, this situation is exacerbated by low energy prices, which contribute to inefficient energy use and a lack of motivation for energy conservation practices among consumers and businesses (Tol and Madessa, 2024; Ghahramani et al., 2020). In contrast, many non-oil-producing countries have implemented stringent regulations aimed at improving energy efficiency in building design and heating systems. These regulations focus on optimising energy use to minimise waste and enhance the sustainability of building operations (Cui, 2024; Stern et al., 2001).

A critical consideration in modern building design is the distribution of heat, which directly impacts both energy efficiency and occupant comfort. Among the various heating systems available, underfloor heating (UFH) and radiator-based heating are frequently studied due to their differing operational

characteristics and efficiency profiles. Recent studies have shown that UFH systems provide more consistent thermal comfort by distributing heat more evenly across the room, significantly reducing temperature stratification — where warm air accumulates near the ceiling, leaving cooler air at lower levels. This stratification is particularly problematic in buildings with high ceilings, as pronounced vertical temperature gradients can lead to discomfort and increased energy costs when occupants adjust thermostats to achieve a suitable environment (Ghahramani et al., 2020; Koca et al., 2014; Tol and Madessa, 2024). For instance, occupants in a 2.7-metre-high room may experience a higher perceived temperature due to reduced thermal stratification, whereas taller rooms could exhibit larger gradients that detract from comfort.

Research indicates that UFH systems can reduce energy consumption by operating at lower temperatures compared to traditional radiator systems. For example, Koca et al. (2024) emphasise that integrating UFH with low-temperature heat pumps can enhance energy efficiency and indoor

air quality (Koca et al., 2014). Furthermore, Haslinger et al. (2021) provide insights into validation techniques used in computational fluid dynamics (CFD) simulations of heating systems, underscoring the importance of accurate modelling in predicting thermal performance (d'Adamo et al., 2021). Understanding and mitigating temperature stratification through effective heating design is crucial for modern buildings, especially as energy costs rise and environmental concerns become more pressing. Therefore, incorporating advanced heating technologies such as UFH can play a significant role in achieving sustainable building practices (Iranzo, 2019; Stern et al., 2001).

Previous studies have focused on improving heating efficiency in residential buildings, particularly in colder climates where heating energy consumption is high. However, the scientific gap this paper addresses lies in the need for a more detailed analysis of temperature distribution across different room heights and heating systems. This study aims to provide a numerical simulation of temperature gradients in rooms with varying ceiling heights (2.7 m, 3.0 m, and 3.3 m), using both radiator and underfloor heating systems, to determine which system achieves superior thermal comfort and energy efficiency. Moreover, this paper intends to fill the gap in recent comparative studies, as many existing studies are either outdated or do not focus on the interaction between room height and heating system performance.

Moreover, the impact of ceiling height on room temperature distribution was investigated at heights of 3.3 m, 3.0 m, and 2.7 m. The findings suggest that reducing ceiling height could improve perceived temperature, as rooms with lower ceilings tend to maintain a more uniform temperature distribution. For instance, occupants in a 2.7-metre-high room may experience a higher perceived temperature due to reduced thermal stratification. Research continues to emphasise that low-temperature heating systems, such as heat pumps, enhance energy savings by minimising efficiency losses and improving thermal comfort (Koca et al., 2014; Cui, 2024).

Recent studies have shown that low-temperature heating systems, when paired with modern heat pumps, offer significant advantages in maintaining indoor comfort while improving energy efficiency. The use of such systems allows for better integration with renewable energy sources, such as solar panels, and reduces reliance on conventional fossil fuels (Tol and Madessa, 2024). Additionally, the combination of underfloor heating with low-temperature systems has been shown to improve indoor air quality and occupant comfort, leading to more widespread adoption in both residential and public buildings (Vijayalaxmi, 2023). Traditional Swedish underfloor heating systems have typically

operated with inlet water temperatures well below 80 °C. In contrast, modern systems generally utilise inlet water temperatures of around 55 °C, reflecting advancements in design and a focus on improved energy efficiency (Tol and Madessa, 2024).

Currently, more than 50 % of newly built homes in Europe feature underfloor heating as the primary heating method (Ghahramani et al., 2020). This trend has also seen growing adoption in North America. Underfloor heating is especially valued for its ability to create a more consistent and comfortable environment through radiant heat, which is widely appreciated in cultures where sitting and sleeping on the floor is common, such as in many Eastern countries.

Recent research further substantiates the benefits of advanced low-temperature and underfloor heating systems in promoting energy efficiency and occupant comfort. Lund et al. (2014) provide a comprehensive review of modern district heating systems, demonstrating how low-temperature heat distribution networks can be integrated with renewable energy sources to achieve significant efficiency gains (Lund et al., 2014). The International Energy Agency (IEA, 2022) also emphasises the critical role of building energy efficiency measures in mitigating climate change, noting that innovations in heating system design are central to reducing overall energy consumption in both residential and commercial sectors (IEA, 2022). Hesarakı et al. (2015) utilised detailed computational fluid dynamics simulations to compare the thermal performance of underfloor heating systems with traditional radiator systems, confirming that UFH systems achieve a more uniform temperature distribution and reduce thermal stratification under varied conditions (Hesarakı et al., 2015). These additional studies collectively reinforce the view that integrating modern heating technologies with robust control mechanisms and renewable energy sources represents a promising pathway toward sustainable building design.

Although previous studies have emphasised the benefits of low-temperature heating systems – especially underfloor heating – in enhancing energy efficiency and reducing temperature stratification, most research has focused on overall energy consumption or performance in colder climates. A detailed analysis of how room height influences heat distribution and subsequent occupant comfort remains underexplored. This study addresses that gap by numerically simulating temperature gradients across various ceiling heights (2.7 m, 3.0 m, and 3.3 m) under both radiator-based and underfloor heating systems. By investigating the interplay between room height and heat transfer efficiency, the research not only fills an important gap in comparative studies but also provides critical insights for optimising heating strategies

in sustainable building design, thereby improving both energy conservation and occupant comfort in diverse architectural settings.

### Methods

This study utilised Computational Fluid Dynamics (CFD) simulations to model temperature distribution in a room heated by two different systems: conventional radiator heating and underfloor heating (UFH). The room dimensions modelled were 3 m × 3 m in width and depth, with ceiling heights of 2.7 m, 3.0 m, and 3.3 m.

### Simulation Setup

The simulations were conducted using Fluent software, with the mesh generated in Gambit. Four distinct mesh densities were tested for independence, and the optimal mesh was selected based on the minimal discrepancy between test cases (as shown in Table 1).

#### Boundary Conditions:

- **Room dimensions:** 3 m × 3 m × 4 m (width × depth × height).
- **Heater dimensions:** 1 m × 0.8 m × 0.5 m.
- **Ceiling heights simulated:** 2.7 m, 3 m, and 3.3 m.
- **Heating system conditions:**
  - Radiator heating was modelled as a heat flux boundary with a constant output of 2kW.
  - Underfloor heating utilised a uniform heat distribution across the floor surface.
  - Ambient temperature: 280 Kelvin (7 °C).
  - Heat transfer coefficient for the walls:  $h = 10 \text{ W/m}^2$ .
  - Wall thickness: 20 cm, with a thermal conductivity of  $K = 1 \text{ W/mK}$  (brick material).

Turbulence was modelled using the RNG k- $\epsilon$  equation, accounting for buoyancy effects. Density changes due to temperature differences were modelled using the ideal gas law, given that temperature differentials in the simulations exceeded 20 °C.

#### Input Parameters:

- **Air properties:** modelled as an ideal gas.
- **Simulation assumptions:** fully turbulent flow, negligible molecular viscosity, heat transfer dominated by convection and radiation.

## Results

### Verification

For validation purposes, our simulation results were compared with established numerical benchmarks reported in recent CFD studies on thermal engineering, rather than with experimental measurements from studies such as Pereira et al. (2009), which do not provide temperature data for the same room configuration. Specifically, our numerical temperature distributions were evaluated against published data from Iranzo (2019) and Haslinger et al. (2021), both of which emphasise the importance of robust validation techniques in CFD applications (Iranzo, 2019; d'Adamo et al., 2021). Although these references do not offer direct experimental data for our specific room geometry, their reported geometrical specifications and boundary conditions served as reliable benchmarks for our simulation setup. Our analysis indicates that the maximum deviation between our simulated temperature profiles and those reported in the literature is within 5 %, thereby confirming the accuracy of our CFD model. This validation demonstrates the effectiveness of our approach in predicting heat transfer performance across different room heights, and it lays the groundwork for future studies that will incorporate additional comparative parameters, such as heat flux and air velocity measurements, to further refine the model's reliability.

To further enhance the robustness of our validation, future studies will incorporate additional comparative parameters, such as heat flux measurements and air velocity profiles. Including these factors will provide a more comprehensive evaluation of the simulation model, ensuring that it accurately captures the complex interactions governing heat transfer and fluid flow in indoor environments. Such thorough validation is essential for developing reliable models that can inform energy-efficient building designs and effective climate control strategies.

Based on the figure depicted below (Fig. 1), it is observed that the maximum deviation of the graph from the data obtained from the paper's chart is within a margin of 5 %. Consequently, the simulation

Table 1. Average surface temperatures and mesh details for different faces (Source: Authors)

| Mesh No.   | Mesh Row | Face 1 Temperature (°C) | Face 2 Temperature (°C) | Face 3 Temperature (°C) | Face 4 Temperature (°C) | Face 5 Temperature (°C) |
|------------|----------|-------------------------|-------------------------|-------------------------|-------------------------|-------------------------|
| 466,253    | 1        | 306.3                   | 316.4                   | 330.6                   | 314.2                   | 310.1                   |
| 864,235    | 2        | 309.4                   | 319.3                   | 338.8                   | 318.5                   | 312.3                   |
| 1,023,268  | 3        | 310.6                   | 321.4                   | 339.1                   | 319.3                   | 313.5                   |
| 15,846,213 | 4        | 310.4                   | 321.7                   | 339.4                   | 319.8                   | 313.3                   |

Notes:

- **Mesh No.:** Total number of elements in the mesh.
- **Mesh Row:** Identifier for the mesh configuration.
- **Face 1-5 Temperatures:** Average surface temperatures of the respective faces, in degrees Celsius (°C).

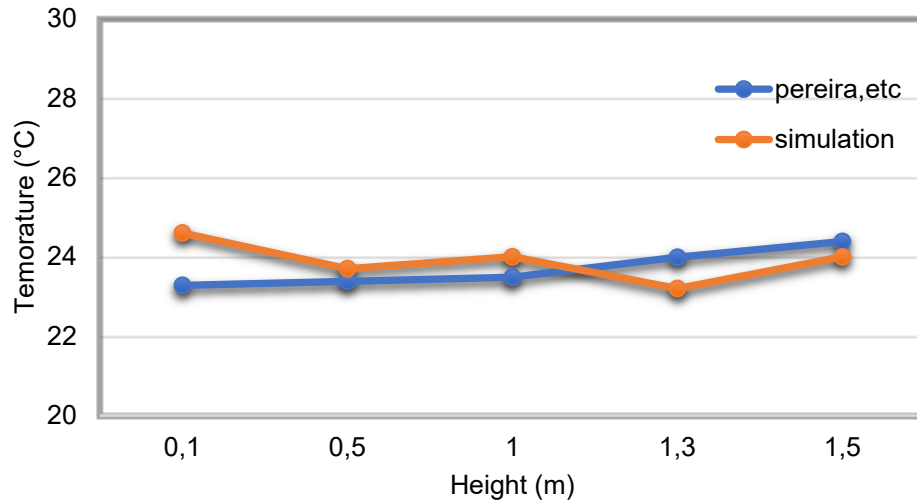


Fig. 1. Simulation verification chart (Source: Authors)

results aligned with those reported in the article are deemed to be satisfactory and reliable.

#### Determining the Boundary Conditions of the Problem

At this juncture, it is essential to define the boundary conditions for the different sections in our Fluent simulation. These conditions govern parameters such as velocity, fluid pressure, and heat transfer processes on the model's walls. In our study, all boundaries are defined as walls and are grouped into three distinct categories:

##### 1. Heater Boundary Condition (Warm Surface):

The heater is modelled as a wall with a constant heat flux. For an average heater operating at approximately 2 kW, a uniform heat flux of **2000 W** is applied across the heater surface.

##### 2. Floor Boundary Condition (Insulated Surface):

The floor is treated as an insulating boundary where no heat transfer occurs. This is implemented by setting the heat flux to  $q = 0 \text{ W/m}^2$ .

##### 3. Wall and Ceiling Boundary Condition (Natural Convection):

The surrounding walls and ceiling are modelled with a natural convection boundary condition. A convective heat transfer coefficient of  $h = 10 \text{ W/m}^2\text{K}$  is used. External walls are exposed to an ambient temperature of  $T_{(\infty)} = 280 \text{ K}$  (approximately  $7 \text{ }^\circ\text{C}$ ), representing heat loss to the outside air. For simplicity, although the natural heat transfer rate can vary (typically ranging from  $5 \text{ }^\circ\text{C}$  to  $25 \text{ }^\circ\text{C}$  depending on conditions), a constant value is assumed uniformly across these surfaces. The walls are assumed to be 20 cm thick, constructed of brick with a thermal conductivity of  $K = 1 \text{ W/mK}$ .

In addition to these boundary conditions, density variations due to temperature fluctuations are modelled using the ideal gas assumption. For buoyancy-driven flow, if the temperature difference

between a wall and the ambient fluid is less than  $20 \text{ }^\circ\text{C}$ , the standard buoyancy equation is applied. For larger temperature differentials (exceeding  $20 \text{ }^\circ\text{C}$ ), the ideal gas assumption is used to avoid significant errors. The simulation also employs the  $k-\epsilon$  RNG turbulence model — with pressure and temperature corrections as well as buoyancy effects — to accurately capture turbulence and thermal stratification.

The final boundary conditions set in our simulation are as follows:

- **Heater:** Constant heat flux of 2000 W applied uniformly.
- **Room Walls and Ceiling:** Natural convection with  $h = 10 \text{ W/m}^2\text{K}$  and exposure to an ambient temperature of 280 K.
- **Floor:** Insulated with zero heat flux ( $q = 0 \text{ W/m}^2$ ).
- **Material Properties:** Room walls are modelled as brick with a thermal conductivity  $K = 1 \text{ W/mK}$ .

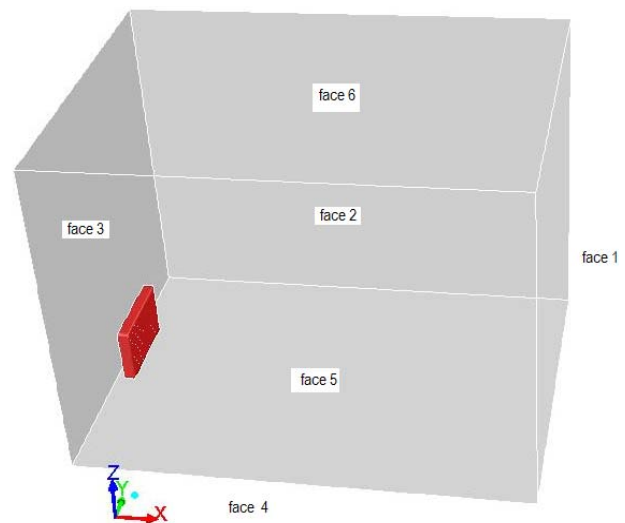


Fig. 2. A view of the simulated room (Source: Authors)

All temperature values in the simulation outputs are reported in degrees Celsius (°C), and unit symbols ( $W$ ,  $m^2K$ ,  $K$ , etc.) have been verified for consistency (Fig. 2).

### Governing Equations

The motion of an incompressible fluid in a turbulent state is described by the Reynolds-averaged Navier–Stokes equations (RANS). These equations consist of the continuity equation, representing mass conservation, and the momentum equation, accounting for the conservation of momentum. The continuity equation is formulated as follows (1):

$$\frac{\partial u_i}{\partial x_i} = 0 \quad i = 1, 2, 3.$$

The momentum equation is:

$$\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + g_i + \frac{\partial}{\partial x_j}(\tau_{ij}) \quad i = 1, 2, 3. \quad (1)$$

In the provided equations,  $u_i$  represents the velocity component in the direction  $x_i$ ,  $P$  denotes the total pressure,  $\rho$  signifies the fluid density,  $g_i$  denotes the acceleration due to gravity in the  $x_i$  direction, and  $\tau_{ij}$  symbolises the stress tensor expressed in the turbulent flow regime, as depicted by the following Equation (2):

$$\tau_{ij} = \left[ p(v + v_t) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] - \left[ \frac{2}{3} p(l + v_t) \frac{\partial u_i}{\partial x_i} \delta_{ij} \right] \quad i = 1, 2, 3. \quad (2)$$

In turbulent flows, shear stress comprises two components. In addition to the shear stress induced by the mean flow component, there exists another shear stress arising from the fluctuating components of velocity, referred to as the Reynolds stresses. This phenomenon is depicted by Equation (3):

$$\tau_{ij} = -\overline{p u'_i u'_j} = p v_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} p k \delta_{ij} \quad i = 1, 2, 3. \quad (3)$$

The kinetic energy of turbulence on the mass unit ( $k$ ) is expressed as follows (4):

$$k = \frac{1}{2} \left( \overline{u'^2} + \overline{u'^2} + \overline{u'^2} \right) \quad \delta_{ij} = \begin{cases} 1 & i = j \\ 0 & i \neq j \end{cases}. \quad (4)$$

The turbulence model is essential for resolving turbulent flow fields based on continuity equations and the Reynolds-averaged Navier-Stokes equations. In this context, within the three-dimensional flow regime, comprising four equations (one continuity equation and three momentum equations), the four unknowns of the flow field — namely velocities in the  $x$ ,  $y$ , and  $z$  directions, and pressure — are determined.

Turbulence models are employed to characterise Reynolds stresses, thereby facilitating closure of the system of equations. One commonly employed model is the standard  $k$ – $\varepsilon$  two-equation model. This model, a semi-empirical method, derives the  $k$  equation precisely from the governing equations of Reynolds-averaged flow. Conversely, the governing equation for  $\varepsilon$  is based on empirical and experimental evidence, as well as mathematical relations. During the derivation of the  $k$  and  $\varepsilon$  equations, it is assumed that the flow is fully turbulent, with the influence of molecular viscosity considered negligible. The equations governing this model are expressed by the following relationships (5, 6):

Equation K:

$$\rho \frac{Dk}{Dt} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + G_k + G_b - \rho \varepsilon. \quad (5)$$

Equation  $\varepsilon$ :

$$\rho \frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_i} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k - G_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k}. \quad (6)$$

The vortex viscosity is determined in the above equations by the following Equation (7):

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}. \quad (7)$$

The constants of the above equations are in accordance with Table 2.

$\delta_k$  &  $\delta_\varepsilon$  are known as the perturbation numbers of Prandtl.

$C_{3\varepsilon}$  is a constant number in  $\varepsilon$  equation that expresses the effect of velocity in the direction of gravity and perpendicular to it (8).

$$C_{3\varepsilon} = \tanh \left| \frac{v}{u} \right|. \quad (8)$$

Hence, when the flow direction aligns with the gravitational direction, the value of the shear layer in this equation equals 1, whereas if the flow direction is perpendicular to the gravitational direction, this value becomes zero. In the provided equations,  $G_b$  represents a term in the  $k$  equation that accounts for the buoyancy effect, as expressed in the following Equation (9):

$$Gk = -\rho \overline{u'_i u'_j} \frac{\partial u_j}{\partial x_i}. \quad (9)$$

The floating effect observed by the term  $G_b$  is effective when there is a constant temperature and gravity in the problem (10).

$$Gb = \beta g_i \frac{\mu_t}{Pr_t} \frac{\partial T}{\partial x_i}. \quad (10)$$

Table 2. Fixed coefficients for standard equations (Source: Authors)

| $C_\mu$ | $C_{1\varepsilon}$ | $C_{2\varepsilon}$ | $\delta_k$ | $\delta_\varepsilon$ |
|---------|--------------------|--------------------|------------|----------------------|
| 0.09    | 1.44               | 1.92               | 1.00       | 1.30                 |

### Investigating the Contours Related to Simulation

In this section, we present a series of snapshots taken at different time instants during the heating process, illustrating the evolution of velocity vectors and temperature contours within the room. It is important to clarify that the figures do not represent distinct spatial sections of the room, but rather show the transient behaviour of the system over time. In our context, the term “fluid” refers to the air within the room. As the air is heated by the heater, its density decreases, causing it to rise (buoyancy-driven flow), while cooler air descends to replace it.

Figure 3 shows velocity vectors captured at six different time instants during the heating process.

These snapshots illustrate the temporal evolution of the airflow within the room.

Figure 4 presents temperature contours obtained at five different time instants, displayed in various vertical orientations. These contours show that as the heated air rises, the temperature increases from the bottom of the room upward.

Figure 5 demonstrates additional temperature contour snapshots that further detail the progression of the temperature field over time.

Figure 6 depicts density distribution at different time instants, highlighting how the reduced density of heated air drives the upward flow.

Figure 7 displays more velocity vector snapshots, providing further insight into the dynamic behaviour of the air movement over time.

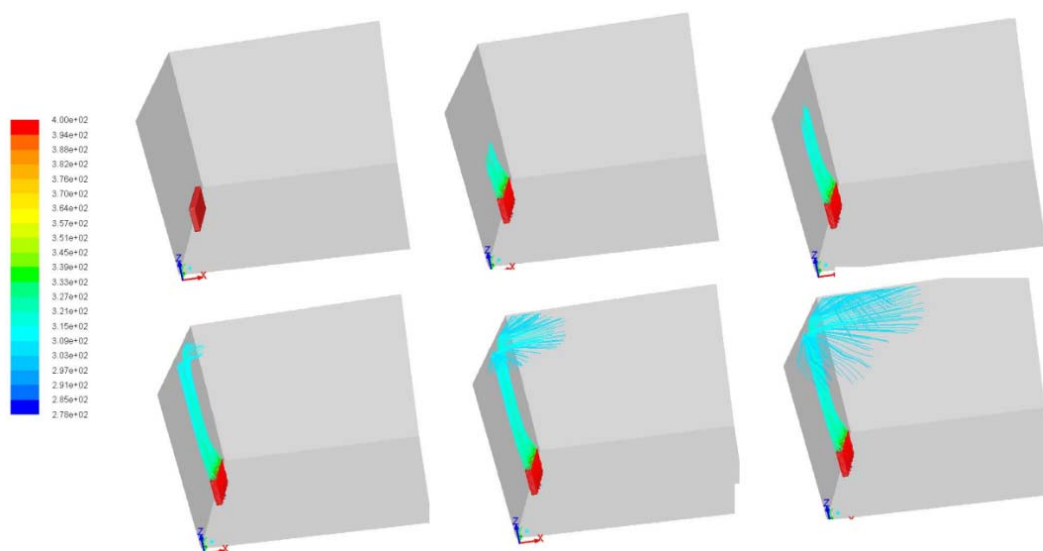


Fig. 3. Speed vector in 6 different sections of the room (Source: Authors)

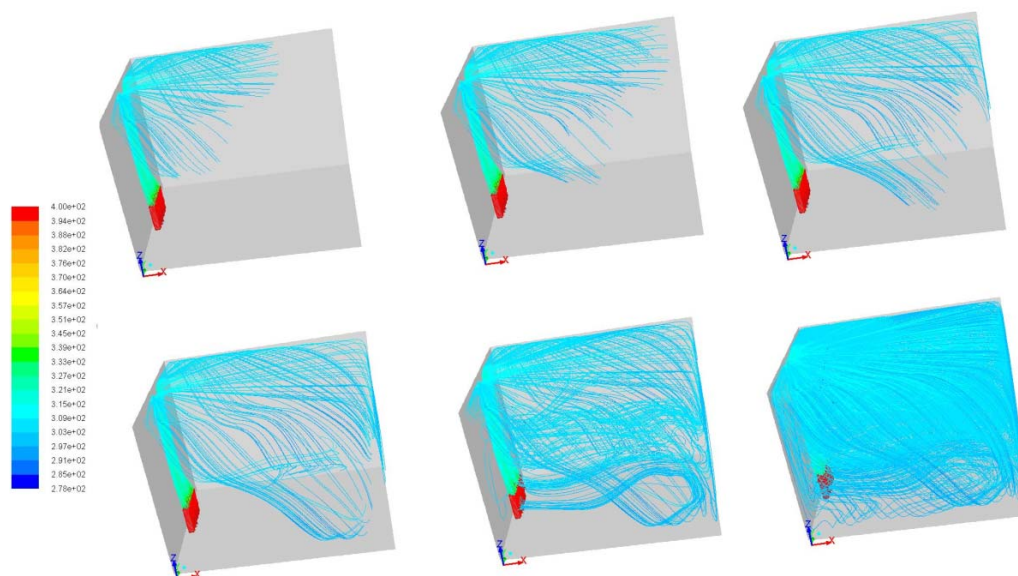


Fig. 4. Temperature contour in 5 different sections in different vertical directions (Source: Authors)

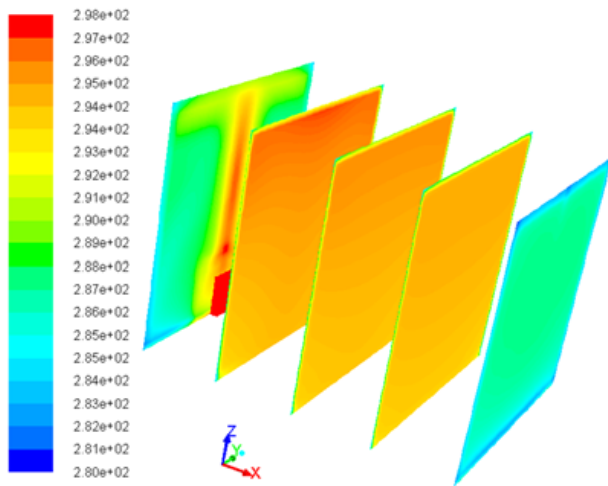


Fig. 5. Temperature contour in 5 different sections of the room  
(Source: Authors)

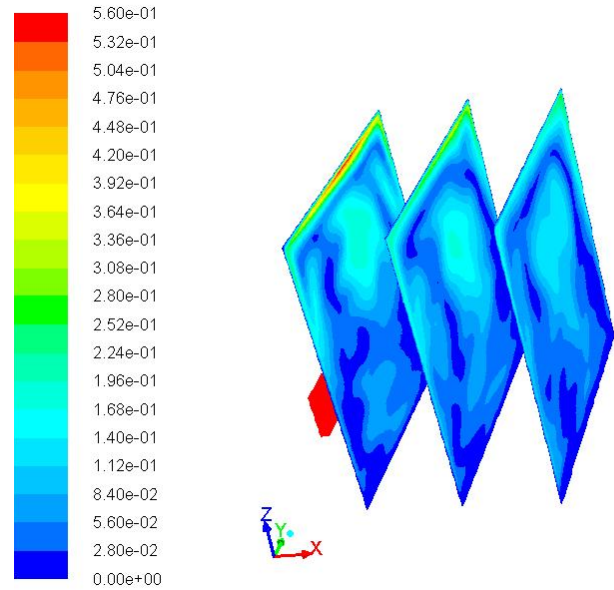


Fig. 8. Speed contour in 5 different sections (Source: Authors)

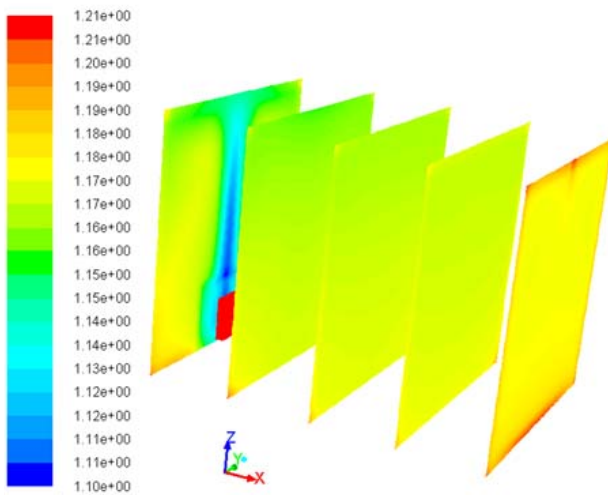


Fig. 6. Density distribution in five different modes (Source: Authors)

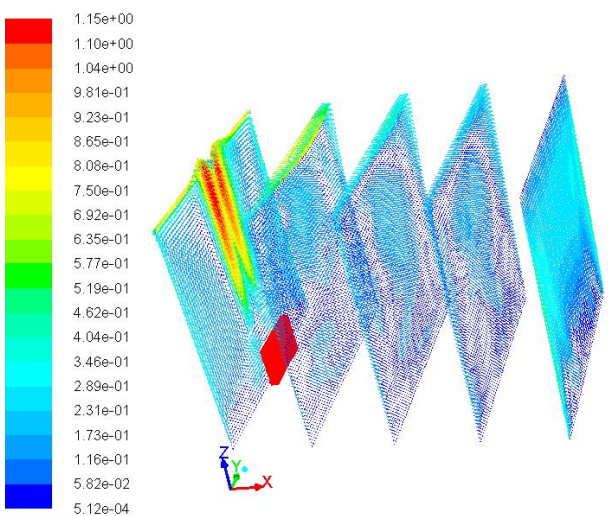


Fig. 7. Velocity vectors in 5 different sections of the room  
(Source: Authors)

Figure 8 illustrates speed contours that quantitatively depict the magnitude of air velocity at various time instants.

The results summarised in Table 3 compare the average surface temperatures at the room faces for different ceiling heights. The updated figures now clearly demonstrate the temporal evolution of the system rather than static, separate sections. This approach allows us to analyse how the air (fluid) responds over time to the heating process.

As shown in the figures, as the heated air ascends, it creates a vertical temperature gradient that evolves over time. This buoyancy-driven movement is essential for understanding how ceiling height influences the overall thermal stratification and, consequently, the thermal comfort within the room. For example, while a room with a 2.7 m ceiling may exhibit higher localised temperatures near occupants due to reduced stratification, a 3.3 m ceiling tends to foster a more uniform temperature distribution across the room (Vijayalaxmi, 2023; Cui, 2024).

Furthermore, to enhance our analysis, we provide average room temperature values and standard deviations, which offer a quantitative measure of the heat distribution uniformity. Figs. 9 and 10 further illustrate these findings by comparing temperature contours across different ceiling heights and contrasting the performance of radiator versus underfloor heating systems. This comprehensive evaluation underscores that underfloor heating achieves superior thermal uniformity, thereby optimising energy efficiency and occupant comfort (Ghahramani et al., 2020).

### Discussion and Conclusion

The results of this study offer several significant insights into how room height influences thermal

Table 3. Average surface temperatures at room faces (°C) and corresponding ceiling heights (m)  
 (Source: Authors)

| Face 5<br>Temperature (°C) | Face 4<br>Temperature (°C) | Face 3<br>Temperature (°C) | Face 2<br>Temperature (°C) | Face 1<br>Temperature (°C) | Ceiling Height<br>(m) |
|----------------------------|----------------------------|----------------------------|----------------------------|----------------------------|-----------------------|
| 329                        | 335                        | 348                        | 345                        | 311.1                      | 2.7                   |
| 313.5                      | 319.3                      | 339.1                      | 321.4                      | 310.6                      | 3.0                   |
| 307                        | 315                        | 332                        | 318                        | 301                        | 3.3                   |

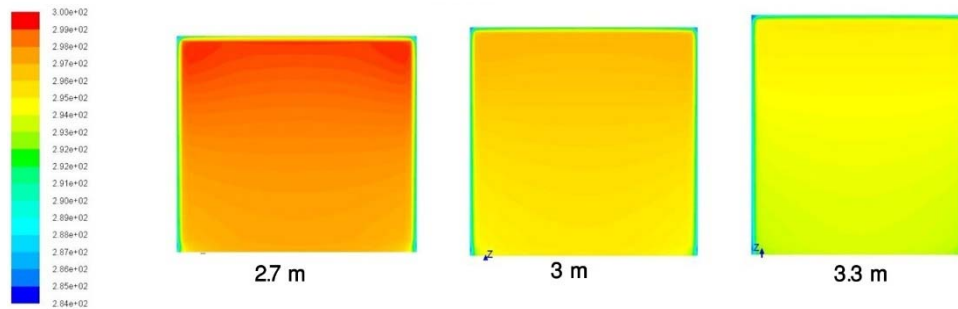


Fig. 9. Distribution of heat in rooms with different ceiling heights (Source: Authors)

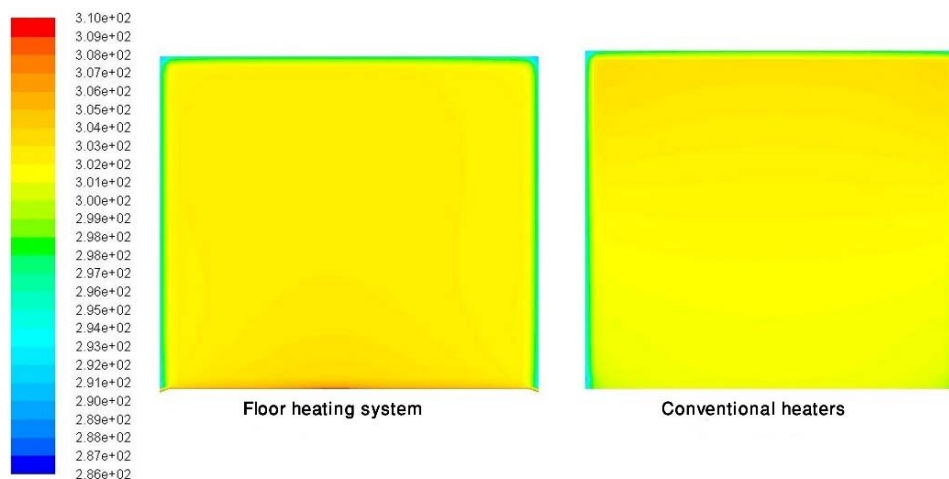


Fig. 10. Distribution of heat with two different heating modes (Source: Authors)

comfort and energy efficiency in buildings using both underfloor heating (UFH) and radiator-based heating systems. To enable a direct comparison, separate CFD simulations were performed for each heating method under identical room conditions and for three distinct ceiling heights (2.7 m, 3.0 m, and 3.3 m). The analysis shows that radiator heating tends to produce significant vertical temperature stratification — particularly in rooms with lower ceilings — where warm air accumulates near the ceiling while cooler air remains near the floor. In contrast, the UFH simulations reveal that underfloor heating delivers a more consistent temperature distribution throughout the occupied zone, effectively reducing the vertical temperature gradient.

Specifically, the simulation data indicate that although a 2.7 m ceiling may yield higher localised

temperatures near occupants for both systems, the overall temperature distribution is markedly more uniform in rooms with a 3.3 m ceiling when using UFH. This is primarily because UFH systems heat the lower part of the room directly, causing a gentle upward flow of air that minimises the pronounced stratification observed with radiator systems. With radiator heating, the convective movement of air tends to create distinct layers: cooler air remains at the lower levels while heated air rapidly rises and accumulates near the ceiling, leading to uneven heating and potential discomfort for occupants. Furthermore, by directly comparing CFD simulation results for both heating systems, our study demonstrates that UFH systems, when integrated with modern low-temperature heat pumps, not only provide superior thermal uniformity but also enhance energy efficiency. The UFH system's ability to

operate at lower water temperatures, typically around 55 °C, reduces energy consumption relative to conventional radiators that require higher operating temperatures. This characteristic makes UFH systems more compatible with renewable energy sources — such as solar thermal or geothermal systems — further contributing to the sustainability of building operations.

The comprehensive analysis includes time-dependent contour plots, velocity vectors, and density distributions, which collectively illustrate the dynamic behaviour of air (the fluid) under both heating scenarios. These detailed visualisations confirm that the interaction between room height and heating system type is a critical factor in achieving optimal thermal performance. For instance, while a room with a 2.7 m ceiling shows higher average temperatures near the floor, the increased stratification in radiator-heated environments can lead to discomfort and inefficient energy use. Conversely, in a 3.3 m room, the UFH system fosters a more gradual temperature increase from floor to ceiling, leading to a balanced indoor environment that enhances occupant comfort.

This study's findings have broad implications for architects and engineers tasked with designing energy-efficient buildings. By demonstrating that the effectiveness of a heating system is not solely dependent on the system's inherent properties but also on the architectural configuration — specifically, room height — our research emphasises the need for integrated design approaches. Such approaches consider both the physical dimensions of a space and the type of heating technology to optimise energy consumption and ensure consistent thermal comfort.

The comparative numerical results presented in this study indicate that underfloor heating (UFH) is likely to produce more uniform temperature

distributions than conventional radiator systems across the ceiling heights considered; however, these findings must be regarded as provisional predictions rather than definitive conclusions. Numerical modelling provides valuable insight into expected behaviours and sensitivities, but real-world applicability can only be established through systematic experimental verification and uncertainty assessment. To address this, we plan to undertake an experimental validation programme that will: (a) reproduce the CFD boundary conditions in instrumented test rooms (or a controlled climatic chamber); (b) acquire steady-state and transient measurements using vertical thermocouple arrays, surface thermistors / IR thermography, local velocity probes, and heat-flux sensors; and (c) compare model outputs with measurements using standard statistical metrics (e.g., mean bias, RMSE) together with uncertainty quantification and sensitivity analysis. If full-scale testing is not immediately feasible, we will begin with scaled laboratory tests and a verification hierarchy (canonical → realistic geometry) to progressively build confidence in the models. We also recommend that future work extend validation to additional variables that influence indoor thermal performance — occupancy patterns, varied construction materials, ventilation / supplemental systems (e.g., ceiling fans), and different control strategies — so that the combined modelling and experimental approach can inform robust, evidence-based design recommendations. By explicitly pairing CFD predictions with a planned programme of targeted experiments, we aim to move from plausible numerical hypotheses to validated knowledge that can reliably guide architects, engineers, and policymakers in selecting heating solutions that balance comfort and energy efficiency.

## References

- (IEA), International Energy Agency. (2022). *Energy Efficiency 2022*, Paris: IEA.
- Cui, G. (2024). Geometric optimisation of the space around the buildings based on the improvement of thermal comfort efficiency of the building. *International Journal of Low-Carbon Technologies*, Vol. 19: pp. 1288–1300.
- d'Adamo, A., Haslinger, M., Corda, G., Höflinger, J., Fontanesi, S., and Lauer, T. (2021). Modelling Methods and Validation Techniques for CFD Simulations of PEM Fuel Cells, *Processes*, Vol. 9, Issue 4: p. 688.
- Ghahramani, A., Galicia, P., Lehrer, D., Varghese, Z., Wang, Z., and Pandit, Y. (2020). Artificial intelligence for efficient thermal comfort systems: Requirements, current applications and future directions, *Frontiers in Built Environment*, Vol. 6: p. 49.
- Hesaraki, A., Bourdakis, E., Ploskić, A., and Holmberg, S. (2015). Experimental study of energy performance in low-temperature hydronic heating systems, *Energy and Buildings*, Vol. 109: pp. 108–14.
- Iranzo, A. (2019). CFD Applications in Energy Engineering Research and Simulation: An Introduction to Published Reviews, *Processes*, Vol. 7, Issue 12: p. 883.
- Koca, A., Gemici, Z., Bedir, K., Böke, E., Kanbur, B.B., and Topaçoğlu, Y. (2014). Thermal Comfort Analysis of Novel Low Exergy Radiant Heating Cooling System and Energy Saving Potential Comparing to Conventional Systems. In *Progress in Exergy, Energy, and the Environment* edited by Dincer, I., Midilli, A., and Kucuk, H. Cham: Springer International Publishing.
- Lund, H., Werner, S., Wiltshire, R., Svendsen, S., Thorsen, J. E., Hvelplund, F., and Mathiesen, B. V. (2014). 4th Generation District Heating (4GDH): Integrating smart thermal grids into future sustainable energy systems, *Energy*, Vol. 68: pp. 1–11.
- Pereira, M. L., Graudenz, G., Tribess, A., and Morawska, L. (2009). Determination of particle concentration in the breathing zone for four different types of office ventilation systems, *Building and Environment*, Vol. 44: pp. 904–11.
- Stern, F., Wilson, R. V., Coleman, H. W., and Paterson, E. G. (2001). Comprehensive approach to verification and validation of CFD simulations – Part 1: Methodology and Procedures, *Journal of Fluids Engineering*, Vol. 123, Issue 4: pp. 793–802.
- Tol, H. İ. and Madessa, H. B. (2024). Return-Temperature Reduction at District Heating Systems: Focus on End-User Sites, *Energies*, Vol. 17, Issue 19: p. 4901.
- Vijayalaxmi, J. (2023). Study of Indoor Thermal Performance Due to Varying Ceiling Heights in a Hot-Humid Climate. In *Building Thermal Performance and Sustainability*, pp. 45–38. Singapore: Springer Nature Singapore.

## ВЛИЯНИЕ ВЫСОТЫ ПОМЕЩЕНИЯ НА ЭФФЕКТИВНОСТЬ ТЕПЛОПЕРЕДАЧИ В РАЗЛИЧНЫХ СИСТЕМАХ ОТОПЛЕНИЯ: ЧИСЛЕННЫЙ АНАЛИЗ

Абдолали Сабети Монфаред<sup>1</sup>, Ахмад Могаддаси<sup>2\*</sup>, Моджтаба Пургхоламали<sup>3</sup>

<sup>1</sup> Кафедра инженерии, компания Omran Abshar Espadana, Тегеран, Иран

<sup>2</sup> Школа дизайна и архитектуры, факультет искусств и дизайна, Университет Канберры, Канберра, АСТ, Австралия

<sup>3</sup> Факультет гражданского строительства, Исламский университет Азад, Наджафабад, Иран

\*E-mail: Ahmad.moghaddasi@canberra.edu.au

**Цель исследования:** настоящее исследование посвящено изучению влияния различной высоты помещений на эффективность теплопередачи систем отопления, с особым акцентом на сравнении традиционного радиаторного отопления и системы напольного отопления. **Методы.** При помощи вычислительной гидродинамики (CFD) были проанализированы распределения температуры при трёх значениях высоты потолка: 2,7 м, 3,0 м и 3,3 м. Численная модель использовалась для исследования характера воздушных потоков, тепловой стратификации и изменения температуры в зоне пребывания людей при различных условиях отопления. **Результаты** показывают, что меньшая высота помещения способствует снижению вертикальной температурной стратификации и, следовательно, повышению общего воспринимаемого уровня температуры, тогда как в более высоких помещениях наблюдаются более выраженные температурные градиенты между полом и потолком. Моделирование также показало, что расположение источника тепла и способ подачи тепловой энергии оказывают существенное влияние на распределение температуры внутри помещения. Кроме того, системы напольного отопления продемонстрировали более высокую тепловую равномерность и энергоэффективность по сравнению с радиаторными системами благодаря более равномерному распределению тепла в нижней, занятой людьми зоне помещения. **Выводы.** Полученные результаты могут способствовать оптимизации энергоэффективности и теплового комфорта пользователей зданий посредством адаптации стратегий отопления к конкретным геометрическим параметрам помещений. Исследование также подчёркивает необходимость учитывать геометрию помещения наряду с выбором системы отопления при оценке тепловых характеристик внутренней среды и разработке энергоэффективных решений для закрытых помещений.

**Ключевые слова:** подогрев пола, теплопередача, естественное вытеснение, вычислительная гидродинамика.