Architectural Studies
International Journal about Architecture

Volume 12, No. 2

UDC 697:536.2:004.94
DOI: 10.56318/as/2.2026.02

ISSN: 2411-801X
e-ISSN: 2786-7374

Murodil Madaliev
Doctor of Sciences in Hydraulic Engineering
Ferghana State Technical University
150109, 86 Ferghana Str., Ferghana, Uzbekistan
https://orcid.org/0000-0001-9994-8907

Mirzokhid Sodikov*
Master
Ferghana State Technical University
150109, 86 Ferghana Str., Ferghana, Uzbekistan
https://orcid.org/0009-0004-0991-6039

A simulation-methods for effective heat energy distribution in heating systems in public buildings

Abstract. The relevance of the research was determined by the existing problem of uneven heat distribution in top-down single-pipe heating systems in buildings, which led to overheating of upper floors and insufficient heating of lower floors, thereby negatively affecting indoor thermal comfort and energy efficiency. The purpose of the work was to analyse methods of improving the indoor microclimate and equalise room temperatures by ensuring uniform distribution of the heat carrier in a top-down designed single-pipe heating system in public buildings. This study used numerical simulation in COMSOL Multiphysics, a CFD software package, to examine the impact of a radiator connection scheme on the heat distribution in the heating system of a five-story public building. The modern vertical single-pipe heating system was examined in the study. System that was managed by valves on the higher floors was created, and the two were contrasted. The local heat flux, flow velocity, and room air temperature were all calculated. Each floor's radiators' temperature loss from the heat transfer medium was assessed independently. The findings demonstrated an obvious unequal heat distribution in the current heating system, with the upper floors overheating and the lower floors not receiving adequate heating. Following the implementation of the new plan, the system's efficiency increased by 21.62%, and the temperature of the heat carrier reaching the first floor increased from 47.813°C to 52.594°C. The practical significance of this study was that it provided an effective method for improving heat distribution in multi-story buildings by regulating flow using a bypass pipe in a single-pipe heating system, which increased thermal comfort and reduced energy consumption

Keywords: heat transfer; CFD-simulation; radiator heating system; energy efficiency; thermal comfort; bypass regulation

INTRODUCTION

Since 2010, enhancing building energy efficiency and improving thermal comfort have become major policy priorities not only in Uzbekistan but also worldwide. The construction sector accounted for nearly 40% of global energy consumption; therefore, improving energy efficiency in buildings remained a key objective in modern engineering science (Global Alliance for Buildings..., 2018). In multi-story buildings with single-pipe heating systems, heat distribution presented a significant challenge, as the temperature of the heat carrier decreased along the vertical

Suggested Citation:
Madaliev, M., & Sodikov, M. (2026). A simulation-methods for effective heat energy distribution in heating systems in public buildings. *Architectural Studies*, 12(2), 19-27. doi: 10.56318/as/2.2026.02.

Journal homepage: https://arch-studies.com.ua/en
Architectural Studies, 12(2), 19-27
Received: 06.02.2026 Revised: 01.05.2026 Accepted: 02.06.2026 Published: 22.06.2026
*Corresponding author (*dr.m.sodikhov@bk.ru*)

direction, leading to overheating on upper floors and insufficient heating on lower floors. Therefore, it was essential to simulate heat transfer processes occurring on indoor surfaces to evaluate building heat balance and improve heating system performance. Convective heat transfer on internal surfaces – such as walls, floors, and ceilings – played a crucial role in indoor air movement, heat transfer, and thermal comfort. ASHRAE (2021) presented information on heat exchange equipment as part of HVAC systems. The section on principles of heating, ventilating, and air conditioning provided essential technical guidance on heat transfer mechanisms across various HVAC configurations. ASHRAE (2022) outlined the ANSI/ASHRAE/IES Standard 90.1-2019, which set energy standards for buildings, excluding low-rise residential buildings. This standard was widely used to ensure that buildings meet energy efficiency requirements for improved sustainability and reduced energy consumption. Heat transfer processes under natural convection conditions exhibited complex hydrodynamic and thermal characteristics, making them difficult to describe accurately using analytical methods. As noted by J.N. Reddy *et al.* (2022), this complexity necessitated the precise determination of convective heat transfer coefficients to improve the reliability of building thermal simulations.

Heat exchange and airflow in enclosed spaces have been extensively studied using computational fluid dynamics (CFD), as reported by N. Serra (2023). However, many CFD models relied on standard wall functions derived from simplified geometries such as pipes or flat plates. As argued by P. Mičko *et al.* (2021), such simplifications may reduce the accuracy of simulations for realistic indoor environments, particularly in representing near-wall turbulence and temperature gradients. Studies by F. Alamdari & G.P. Hammond (1983) proposed empirical correlations for convective heat transfer coefficients under natural convection conditions, which remained widely used in building thermal modelling. More recent studies, such as M. Camci *et al.* (2023), have developed improved correlation models based on experimental and numerical investigations of convection on building surfaces. M. Singh & R.A. Sharston (2022) demonstrated that integrating adaptive convection algorithms into building energy simulation tools significantly improved prediction accuracy compared to conventional constant-coefficient approaches. Furthermore, integrating CFD methods with building energy models enabled a more comprehensive analysis of indoor heat transfer processes. P.V. Nielsen (2015) had also emphasised the importance of accurate modelling and optimisation of heating systems to improve indoor thermal conditions and reduce energy consumption. In particular, Y. Li *et al.* (2024) showed that coupling CFD with building simulation enhanced control strategies and overall system efficiency. Despite these advances, the analysis of vertical single-pipe heating systems in multi-story buildings remained insufficiently explored, particularly in terms of heat distribution along the building height. Therefore, this study aimed to investigate heat distribution in a multi-story building with a vertical single-pipe heating system using CFD modelling, with focus on flow redistribution and its impact on thermal conditions across different floors.

MATERIALS AND METHODS

The study examined the thermal conditions of a typical classroom in the educational building of the Faculty of Architecture and Construction at Fergana State Technical University, located in Fergana, Uzbekistan. The building was a five-story reinforced concrete structure, with a total floor area of 2,963 m^2 and a volume of 8,517.45 m^3, typical for public educational buildings in the region. The building had a maximum heating load of 357,752 W, a heating load density of 120.75 W/m^2, and was characteristic of buildings in the region. In order to determine the thermal boundary conditions and indoor climate parameters, the study used the recommendations of EN 16798-1:2019 (2019) and ASHRAE (2022). National normative documents were used, such as SNK 2.08.02-23 (2024) and SNK 2.01.18-24 (2025). Numerical simulation of the flow and heat transfer in the room was carried out using the COMSOL Multiphysics software. In this regard, the continuity, momentum, and energy equations were solved using the Reynolds Averaged Navier Stokes (RANS) method along with the Standard k-epsilon model. Heat flux was determined (Kreider 2009) by the formula:

$$q = h(T_s - T_a), \quad (1)$$

where h – heat transfer coefficient (W/(m^2·K)), T_s – surface temperature (°C), T_a – air temperature (°C). Nusselt number was determined by the formula:

$$Nu = \frac{hL}{k}, \quad (2)$$

where h – heat transfer, L – characteristic length, k – thermal conductivity (W/(m·K)). Reynolds number was determined by the formula:

$$Re = \frac{\rho v L}{\mu}, \quad (3)$$

where ρ – density (kg/m^3), v – velosity (m/s), μ – dynamic viscosity (Pa·s).

The presented equations were used to evaluate the heat transfer characteristics of the radiator. The Reynolds number was calculated to determine the flow regime of the heat carrier, confirming turbulent flow conditions within the system. Based on this, the Nusselt number was obtained, which allowed the calculation of the convective heat transfer coefficient. Using this coefficient, the heat flux between the radiator surface and the surrounding air was determined. These calculated parameters were subsequently used to analyze the temperature distribution and assess the thermal performance of the radiator system. This model was used due to its stability and efficiency in simulating the flow in indoor environments. At the same time, the heat transfer in fluids and turbulent flow interfaces in COMSOL have been used simultaneously to solve the heat transfer and fluid flow problems in the room. In the computational domain, a three-dimensional rectangular room was created. In the room, one radiator and one window were placed. At the boundaries, the surface temperature of the radiator was taken as 348 K, while that of the window was taken as 268 K. At the same time, the walls, floors, and ceilings of the room were taken at a temperature of 293 K (Fig. 1).

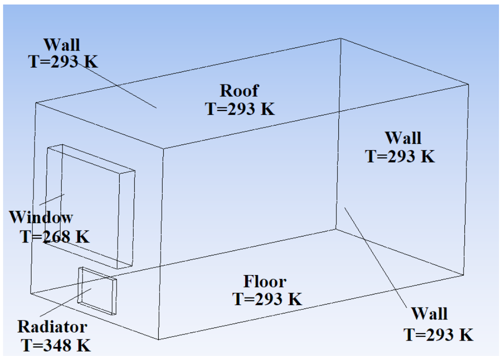

Figure 1. Computational domain and boundary conditions for the modeled room with radiator heating system
Source: developed by the authors

A computational mesh was created using COMSOL. In this regard, a high-resolution mesh was created in the areas close to the radiator and window. Also, in study were obtained the temperature and humidity data. The simulated values of heat carrier inlet and outlet temperatures, mean radiator temperatures, and indoor air temperatures were extracted for each floor and orientation (north and south) of the building, covering 30 measurement points across five floors. Bypass pipes equipped with valve regulation mechanisms were installed at the radiators' inlet and outlet junctions on the fifth, fourth, third, and second floors in the suggested configuration. In this study, bypass lines were implemented as alternative flow paths to regulate the distribution of the heat carrier. Instead of being completely discharged through the radiator, a predetermined portion of the heat carrier circulating through the upper floors was diverted downward via the bypass lines to the lower floors by carefully adjusting the valves built into these bypass pipes. This configuration was designed to improve thermal balance by redistributing the flow between the upper and lower floors.

RESULTS AND DISCUSSION

The educational building of the Faculty of Architecture and Construction at Fergana State Technical University's heat supply system used a vertical top-to-bottom connection scheme. In this setup, a single-pipe distribution network was used to supply the heat carrier water to the system from the topmost (fifth) floor at an initial temperature of 70°C. The water then sequentially passed through the radiators placed on each floor. The system's radiators all had the same 840 W thermal capacity and were designed to use radiative and convective mechanisms to transfer heat from the circulating water to the indoor air. The Republic of Uzbekistan's regulatory document SanPiN 0341-16 (2016) stated that maintaining air temperatures in classrooms between 18°C and 21°C was advised for the best thermal comfort. However, real-world observations and simulation results have revealed notable departures from these regulations (Decree of the President..., 2025). In the current system, the lower floors, particularly the first and second degrees, experience a temperature drop to about 16°C, which was below the allowable comfort limit, while the upper floors' interior temperatures reached 23-24°C, resulting in an overheating state. The building's occupants experience inconvenience due to this heat imbalance (Fig. 2), which also degraded the indoor microclimate (EN 16798-1:2019, 2019).

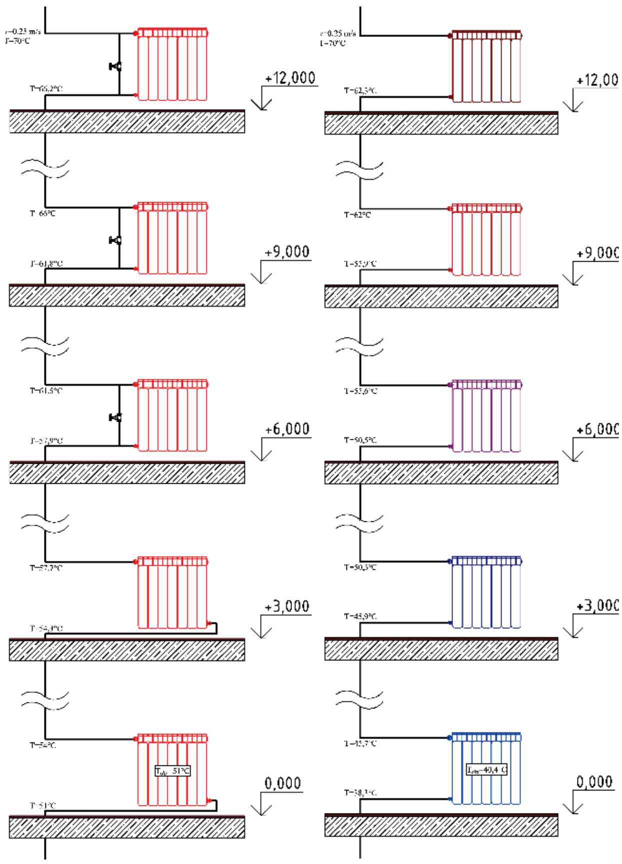

Figure 2. Schematic diagram of the heating system
Note: a – current system; b – the proposed system
Source: developed by the authors

The heat carrier's slow cooling as it descended the vertical riser was the primary cause of this drawback. The water temperature dropped considerably before it reached the lower heating devices because the majority of the thermal energy was released in the upper floors. Consequently, the upper floors received an excess of heat, while the lower floors were deprived of adequate heating capacity. The authors S.V. Kim & R.D. Brown (2021) noted that this unequal heat distribution led to a crisis in the heating system's overall efficiency and served as the foundation for irrational energy resource use. The heat carrier reached the fifth-floor radiator at a temperature of 70°C, according to experimental and numerical research. At its outlet, the liquid's temperature dropped to 64.4°C after traveling through the radiator and transferring heat to the room's air (Table 1).

Table 1. Radiator inlet and outlet temperatures for each floor (existing system)

Floor	Radiator entry T,°C	Radiator exit T,°C	Radiator average ΔT,°C
5th floor	70.000	64.400	5.600
4th floor	64.213	59.453	4.760
3th floor	59.262	55.184	4.078
2th floor	54.993	51.247	3.746
1th floor	51.056	47.813	3.243

Source: developed by the authors

The radiators on the upper floors (the fourth and fifth floors) released a lot of heat. There was a noticeable drop in the value of ΔT as the layers descend. The water temperature coming from the radiator on the first floor was only 47.8°C, which was insufficient to keep the building's comfort levels at their ideal levels. Furthermore, it was discovered that the radiator on the first floor had an average surface temperature of 45°C, which was much lower than the readings on the upper floors (Carella *et al.*, 2023). This redistribution results in the thermal energy that was previously used to create excess temperatures on the upper floors being effectively used to support the heating of the lower floors (Huang *et al.*, 2024). This was more than just a reallocation of flow; it was the recovery and reassignment of thermal energy that would otherwise cause upper-floor spaces to overheat, while lower floors were left insufficiently heated. The outlet temperature on the first floor had increased significantly in comparison to the current baseline configuration; it was previously recorded at 47.813°C, but it had reached 52.594°C, an improvement of about 4.8°C. The system's capacity to provide enough heat to the source was demonstrated by the radiator network's maximum temperature of 55.850°C (Table 2), which was recorded on the upper floors.

Table 2. Temperature difference in radiator

Floor	Radiator entry T,°C	Radiator exit T,°C	Temperature difference in radiator (°C)
5th floor	70.000	66.576	3.424
4th floor	66.385	62.668	3.717
3th floor	62.477	59.908	2.569
2th floor	59.717	56.034	3.683
1th floor	55.850	52.594	3.249

Source: developed by the authors

The building's overall level of comfort had increased as a result of this thermal rebalancing, which had partially mitigated the upper floors' excessive heating and more evenly distributed heat downward (Fanger, 1970). The temperature distribution (°C) of the heat carrier inside the fifth-floor radiator was shown in Figure 3, a. Around 55°C was the temperature, at which the heat carrier entered the radiator through the upper inlet. As it passed through the radiator's internal channels, it progressively transferred thermal energy to the wall surfaces through convective and conductive mechanisms (Incropera *et al.*, 2006). The fluid's temperature had decreased to about 40°C by the time it reached the outlet at the bottom of the radiator, indicating an efficient thermal exchange of about 15°C over the radiator's length under normal operating circumstances. The figure's colour gradient illustrated the degree of heat exchange in the radiator's various zones, showing that thermal transfer was relatively more prominent in the centre.

To provide a quantitative justification of the heat transfer processes, key dimensionless and thermophysical parameters were determined. The Reynolds number was calculated using typical properties of the heat carrier ($\rho \approx 980$ kg/m^3, $\mu \approx 0.00047$ Pa·s at ~60°C), an average flow velocity of $v \approx 0.3$ m/s, and a characteristic length of $L \approx 0.02$ m. Accordingly, the value $Re \approx (980\times0.3\times0.02)/0.00047 \approx 12{,}500$ was obtained. This value significantly exceeded the critical threshold of 4,000, confirming a turbulent flow regime and aligning with the CFD results. Based on these data, the Nusselt number was estimated. For water with a thermal conductivity of $k \approx 0.65$ W/(m·K) and a characteristic length of $L \approx 0.02$ m, assuming a convective heat transfer coefficient in the range of $h = 300 - 600$ W/(m^2·K), the estimated value was $Nu \approx (400\times0.02)/0.65 \approx 12.3$. The obtained values of the convective heat transfer coefficient (h) confirmed this trend: in the baseline configuration, it decreased along the vertical riser, whereas in the modified system it became more uniform. The calculation of heat flux (q) demonstrated that, in the existing system, thermal energy was predominantly concentrated on the upper floors, leading to overheating, whereas after modification a more even distribution was achieved. The consistency between the calculated parameters and the temperature data confirmed a reduction in the temperature gradient (from 22°C to 17°C) and an increase in overall system efficiency by 21.62%, indicating an improvement in heat transfer performance due to flow redistribution.

The cross-sectional area of the fins (green line) and the temperatures in the radiator's lower horizontal pipe (blue line) were contrasted in Figure 3, b. The graph illustrated how the temperature dropped as the heat carrier moves through the pipe, with local maximum values observed on the fins' surface. After implementation of the proposed bypass system, the temperature distribution in the 5th-floor radiator became more uniform, as shown in Figure 3, c. This is further confirmed by the temperature variation along the radiator length (Fig. 3, d), where a more gradual temperature gradient is observed compared to the initial case. As shown in Figure 3, e, the temperature in the radiator outlet pipe decreased to 47.8°C by the first floor, while the velocity field in the 5th-floor radiator (Fig. 3, f) indicates improved flow distribution due to the bypass configuration. The radiator's average temperature dropped to 41-42°C. The device's heating efficiency was reduced as a result of this temperature drop (Carella *et al.*, 2023).

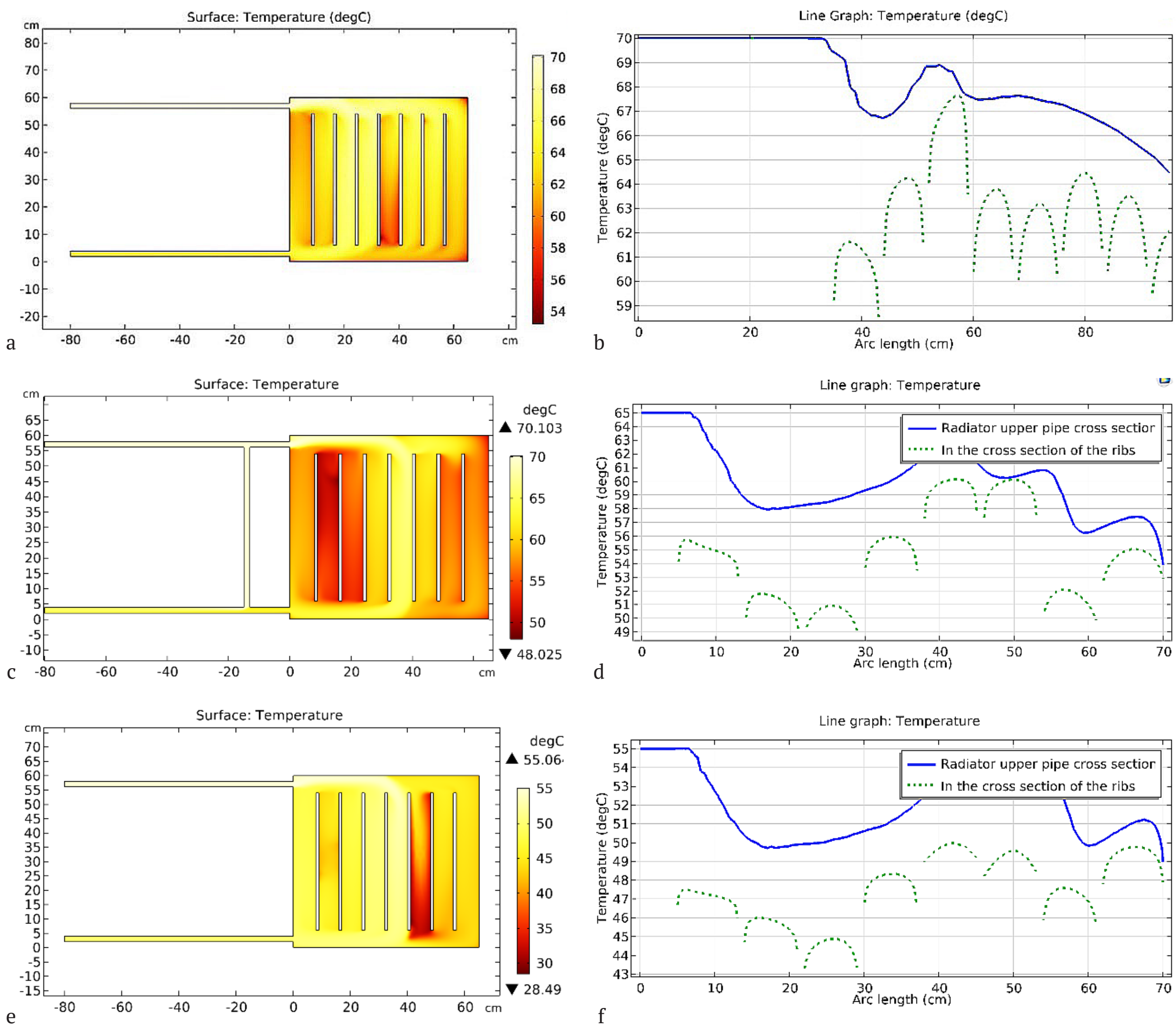

Figure 3. Simulation results for the radiator thermal and flow characteristics

Note: a – temperature distribution of the heat transfer fluid in the 5th-floor radiator (baseline system); b – line graph of temperature variation along the radiator length (baseline system); c – temperature distribution in the 5th-floor radiator after implementation of the proposed bypass system; d – line graph of temperature variation along the radiator length after implementation of the bypass system; e – temperature distribution in the 1st-floor radiator (proposed bypass system); f – velocity field of the 5th-floor radiator in the proposed bypass system

Source: SNK 2.08.02-23 (2024), SNK 2.01.18-24 (2025)

The bypass flow joined the horizontal outlet stream coming from the radiator (approximately 62-63°C, dark yellow zone) after descending vertically through the upper channel (approximately 63-70°C, light yellow zone). A localised turbulence zone forms at the point, where these two streams converge, and it was easily identifiable in the area, where vector density increased and flow directions suddenly changed course (Raftery *et al.*, 2024). The thermal dynamics controlling this process were further supported by the temperature profile of the combined coolant at the bypass mixing junction along the arc length. The temperature stayed relatively constant at about 62°C for the first 0-0.5 cm segment, suggesting a preliminary zone, where the two streams have not yet reached full intermixing. The temperature risen quickly to about 69.5°C within the 0.5 to 1.5 cm range, indicating strong thermal interaction between the convergent fluid streams. The temperature stabilised between roughly 69.5°C and 70°C after crossing the 1.5 cm threshold, indicating that the combined flow had reached a state of complete mixing and thermodynamic equilibrium, fully compliant with the basic law of conservation of energy (Alamdari & Hammond, 1983). Simulation results using CFD showed that there was a considerable thermal imbalance in the existing single pipe system. Under normal operation, the heat transfer fluid entered the fifth-floor radiator at about 70°C, gradually cooling down as it travelled down to the first floor, where it entered the radiator at a considerably lower temperature. Thus, the maximum internal temperature of the fifth-floor radiator was found to be 55.85°C, while the outlet temperature of the first-floor radiator was found to be only 47.81°C, a difference of 8.04°C, which caused overheating of the floors above and insufficient heating of floors below. With the installation of the suggested system of circulation control, the heat carrier was partially rerouted through bypass junctions installed at each floor, thus facilitating uniform heat load redistribution in the vertical circuit. It was defined a notable increase in the radiator outlet temperature on the first floor, from 47.81°C to 52.59°C, a difference of 4.78°C, which was the least favoured portion of the system. The ratio of the actual heat output to the theoretically maximum heat transfer under specific supply conditions was used to evaluate the efficacy of the heat supply system (Incropera *et al.*, 2006). The system's overall efficiency improved by 21.62% points, from 67.14% in the current configuration to 78.38% in the proposed system (Fig. 4).

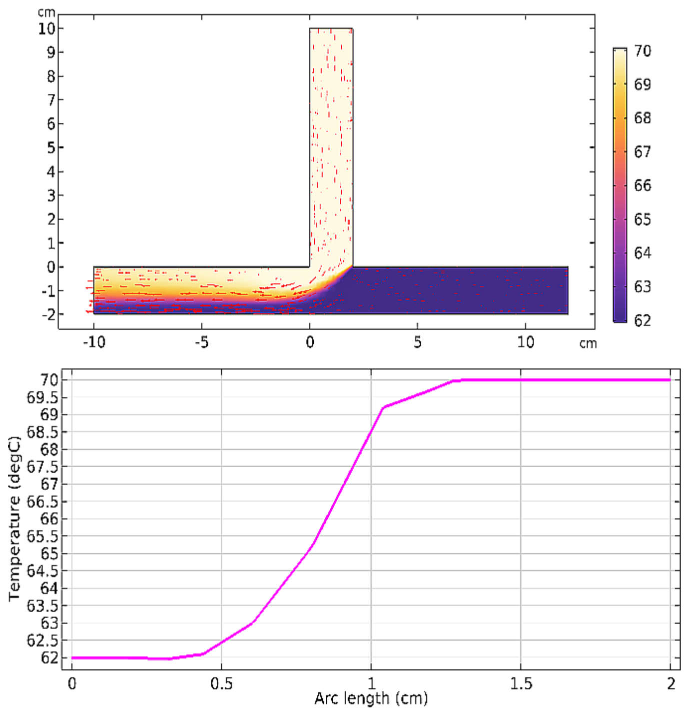

Figure 4. Thermal analysis of the rotary mixing junction

Note: a – the heat flow distribution model with the flow structure and velocity field at the rotary mixing junction; b – the temperature profile of the mixed coolant along the arc length at the bypass mixing junction

Source: developed by the authors

Due to unequal distribution and high exhaust temperatures on the upper floors, this growth demonstrated that a sizable amount of the thermal energy presented in the heat carrier was effectively transferred to building spaces without being wasted. Furthermore, as Figure 5 illustrated, the temperature loss along the radiator network dropped by 5°C from roughly 22°C in the existing system to roughly 17°C in the suggested configuration. Because of this reduction in heat loss, the heat carrier was able to move through the system with greater energy retention, allowing for more effective use at the layer level. The redistribution of the circulating flow, which prevented excessive heat, was directly linked to the temperature drop. Release at any point in the circuit and maintains a more favourable temperature gradient throughout the entire network (Patankar, 1980).

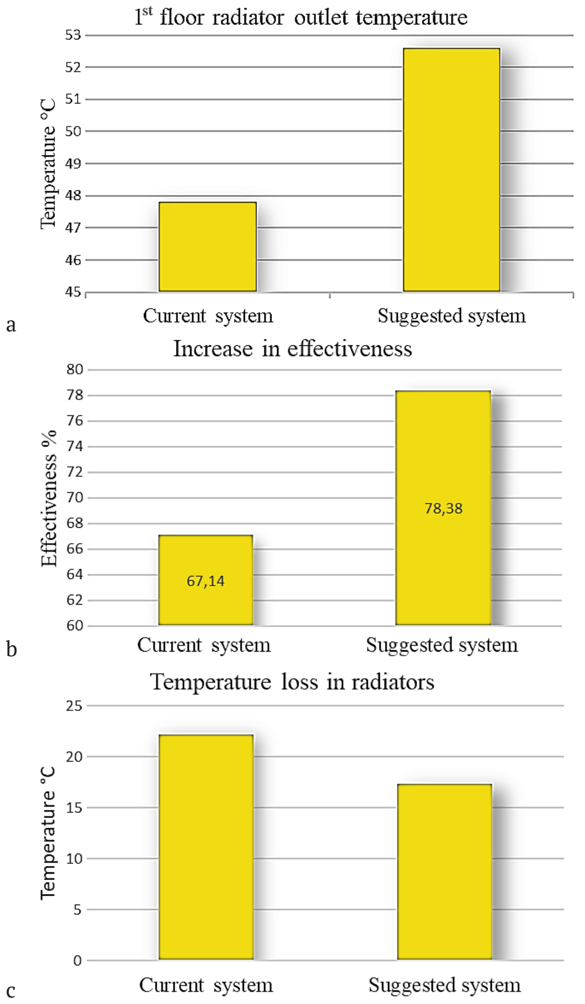

Figure 5. Comparison of current and suggested heating systems

Note: a – 1st floor radiator outlet temperature (°C); b – system heating effectiveness (%); c – temperature loss in radiators (°C)

Source: developed by the authors

The physical coherence of the suggested design solution was further supported by the velocity field analysis at the bypass mixing junction. E. Guelpa *et al.* (2023) noted that a localised turbulence zone formed at the junction of the bypass and radiator outlet streams, increasing convective mixing and hastening thermal equilibration between the two merging streams. The proposed bypass connection arrangement's hydrodynamic effectiveness and geometric efficiency were demonstrated by the temperature profile traced along the arc length at the mixing junction, which confirmed that full thermal equilibrium was reached within about 1.5 cm downstream of the junction. Consequently, compliance with established trends in thermal modelling of premises and optimisation of heating systems was confirmed. Specifically, the increase in radiator outlet temperature on the first floor, from 47.81°C to 52.59°C, was consistent with the results reported by, where the authors demonstrated that using CFD analysis of radiator geometry directly contributed to enhancing the precision of convective heat transfer predictions. The reduction of temperature loss between floors was consistent with the energy redistribution concept presented by M. Singh & R.A. Sharston (2022), where the authors emphasised that using CFD together with building energy models provided a more accurate description of heat transfer within buildings. Finally, the increase in thermal uniformity using bypass regulation was consistent with the study presented by I. Sarbu *et al.* (2022), where researchers reported considerable energy savings using partial redirection of flow in high-rise building heating systems. B.A. Aljashaami *et al.* (2024) conducted a systematic review of recent heating, ventilation, and air conditioning technologies based on renewable energy, substantiating their key role in achieving zero-energy buildings. The authors demonstrated that the integration of hybrid energy systems and intelligent control was a decisive factor in improving energy efficiency and reducing the carbon footprint of buildings. I. Bilous *et al.* (2025) developed a dynamic model for assessing energy efficiency and indoor climatic conditions, enabling a comprehensive analysis of thermal processes in buildings. The study showed that a systemic modelling approach provided more accurate optimisation of indoor environment parameters and energy consumption compared to static methods. Overall, the simulation results demonstrated that the proposed flow redistribution strategy significantly improved heat distribution across all floors. The thermal imbalance observed in the original system was reduced, particularly by enhancing heating conditions on the lower floors. These findings confirmed the effectiveness of the implemented regulation approach in achieving more uniform indoor thermal conditions.

CONCLUSIONS

The results of the CFD simulation performed in COMSOL Multiphysics verified that the existing series-flow configuration led to overheating of the building's floors. The outlet temperature of the first-floor radiator was only 47.813°C, while the internal temperature of the fifth-floor radiator was 55.85°C, resulting in a thermal imbalance of 8.04°C. To resolve the above imbalance without replacing the entire system, a bypass regulation system was suggested, in which bypass pipes would be installed at the radiator connections on floors 2, 3, 4, and 5. Numerical simulation of the system showed a substantial improvement in heat distribution across the floors. The outlet temperature of the radiator on the first floor increased from 47.813°C to 52.594°C, an increase of 4.78°C.

This was the most disadvantaged area. The overall system heating effectiveness improved by 21.62%, from 67.14% to 78.38%. The total temperature loss across the heating network decreased from 22°C to 17°C. Analysis of velocity fields at the bypass mixing junction showed that full thermal equilibrium between the merging flows was attained within approximately 1.5 cm downstream of the junction, thus confirming the hydrodynamic efficiency of the proposed design. The proposed bypass method provided a cost-effective, structurally non-intrusive approach to enhancing energy efficiency in existing single-pipe heating systems. Future research directions could include experimental verification of the method under real operating conditions and optimisation of valve settings for various seasonal/occupancy scenarios.

ACKNOWLEDGEMENTS

The authors express their gratitude to the Faculty of Architecture and Construction at Fergana State Technical University for providing access to the laboratory facilities used in this study.

FUNDING

None.

CONFLICT OF INTEREST

None.

REFERENCES

[1] Alamdari, F., & Hammond, G.P. (1983). Improved data correlations for buoyancy-driven convection in rooms. *Building Services Engineering Research & Technology*, 4(3), 106-112. doi: 10.1177/014362448300400304.

[2] Aljashaami, B.A., Ali, B.M., Salih, S.A., Alwan, N.T., Majeed, M.H., Ali, O.M., Alomar, O.R., Velkin, V.I., & Shcheklein, S.E. (2024). Recent improvements to heating, ventilation, and cooling technologies for buildings based on renewable energy to achieve zero-energy buildings: A systematic review. *Results in Engineering*, 23, article number 102769. doi: 10.1016/j.rineng.2024.102769.

[3] ASHRAE. (2021). Heat exchange equipment. In *Principles of heating, ventilating, and air conditioning*. Atlanta: American Society of Heating, Refrigerating and Air-Conditioning Engineers.

[4] ASHRAE. (2022). *ANSI/ASHRAE/IES Standard 90.1-2019. Energy standard for buildings except low-rise residential buildings*. Retrieved from https://surl.lu/bpgeoq.

[5] Bilous, I., Biriukov, D., Eutukhova, T., Novoseltsev, O., Voloshchuk, V., & Prozuments, A. (2025). Energy efficiency and indoor climatic conditions in buildings: Dynamic modeling for systemic improvement. *Frontiers in Built Environment*, 10, article number 1502794. doi: 10.3389/fbuil.2024.1502794.

[6] Camci, M., Karakoyun, Y., Acikgoz, O., & Dalkilic, A.S. (2023). Case studies on local mixed convection heat transfer coefficients along a radiant heated wall subjected to displacement ventilation. *Journal of Thermal Analysis and Calorimetry*, 148, 7845-7854. doi: 10.1007/s10973-023-12262-6.

[7] Carella, A., Del Ferraro, L., & D'Orazio, A. (2023). Air/water heat pumps in existing heating and hot water systems for better urban air quality and primary energy savings: Scenarios of two Italian cities. *Energies*, 16(1), article number 377. doi: 10.3390/en16010377.

[8] Decree of the President of the Republic of Uzbekistan No. UP-63 "On Measures to Improve Public Administration in the Field of Energy Efficiency Enhancement and the Development of the Energy Service Company Market". (2025, March). Retrieved from https://lex.uz/en/docs/7464127.

[9] EN 16798-1:2019. (2019). *Energy performance of buildings – ventilation for buildings – Part 1: Indoor environmental input parameters for design and assessment of energy performance of buildings addressing indoor air quality, thermal environment, lighting and acoustics – module M1-6*. Retrieved from https://surl.li/edjyuo.

[10] Fanger, P.O. (1970). *Thermal comfort. Analysis and applications in environmental engineering*. Copenhagen: Danish Technical Press.

[11] Global Alliance for Buildings and Construction. (2018). 2018 Global Status Report. Towards a zero-emission, efficient and resilient buildings and construction sector. *United Nations Environment Programme*. Retrieved from https://iea.blob.core.windows.net/assets/30a31866-26ed-4c57-9d6f-3d382442b3fb/2018_Global_Status_Report.pdf.

[12] Guelpa, E., Capone, M., Sciacovelli, A., Vasset, N., Baviere, R., & Verda, V. (2023). Reduction of supply temperature in existing district heating: A review of strategies and implementations. *Energy*, 262, article number 125363. doi: 10.1016/j.energy.2022.125363.

[13] Huang, C., Li, N., He, S., Weng, X., Shu, Y., Yang, G., & Bao, Y. (2024). Investigation of the temperature distribution and energy consumption of an integrated carbon fiber paper-embedded electric-heated floor. *Buildings*, 14(7), article number 2097. doi: 10.3390/buildings14072097.

[14] Incropera, F.P., DeWitt, D.P., Bergman, T.L., & Lavine, A.S. (2006). *Introduction to heat transfer* (5th ed.). Hoboken: Wiley.

[15] Kim, S.W., & Brown, R.D. (2021). Urban heat island (UHI) variations within a city boundary: A systematic literature review. *Renewable and Sustainable Energy Reviews*, 148, article number 111256. doi: 10.1016/j.rser.2021.111256.

[16] Kreider, J.F., Curtiss, P.S., & Rabl, A. (2009). *Heating and cooling of buildings: Design for efficiency* (2nd ed.). Boca Raton: CRC Press.

[17] Li, Y., Li, L., Cui, X., & Shen, P. (2024). Coupled building simulation and CFD for real-time window and HVAC control in sports space. *Journal of Building Engineering*, 97, article number 110731. doi: 10.1016/j.jobe.2024.110731.
[18] Mičko, P., Kapjor, A., Holubčík, M., & Hečko, D. (2021). Experimental verification of CFD simulation when evaluating the operative temperature and mean radiation temperature for radiator heating and floor heating. *Processes*, 9(6), article number 1041. doi: 10.3390/pr9061041.
[19] Nielsen, P.V. (2015). Fifty years of CFD for room air distribution. *Building and Environment*, 91, 78-90. doi: 10.1016/j.buildenv.2015.02.035.
[20] Patankar, S.V. (1980). *Numerical heat transfer and fluid flow*. New York: McGraw Hill Book.
[21] Raftery, P., Singla, R., Cheng, H., & Paliaga, G. (2024). Insights from hydronic heating systems in 259 commercial buildings. *Energy and Buildings*, 321, article number 114543. doi: 10.1016/j.enbuild.2024.114543.
[22] Reddy, J.N., Anand, N.K., & Roy, P. (2022). *Finite element and finite volume methods for heat transfer and fluid dynamics*. Cambridge: Cambridge University Press.
[23] SanPiN 0341-16. (2016). *Sanitary and epidemiological requirements for conditions and organization of education in general secondary schools*. Retrieved from https://surl.li/qsdaab.
[24] Sarbu, I., Mirza, M., & Muntean, D. (2022). Integration of renewable energy sources into low-temperature district heating systems: A review. *Energies*, 15(18), article number 6523. doi: 10.3390/en15186523.
[25] Serra, N. (2023). Revisiting RANS turbulence modelling used in built-environment CFD simulations. *Building and Environment*, 237, article number 110333. doi: 10.1016/j.buildenv.2023.110333.
[26] Singh, M., & Sharston, R.A. (2022). A literature review of building energy simulation and computational fluid dynamics co-simulation strategies and its implications on the accuracy of energy predictions. *Building Services Engineering Research & Technology*, 43(1), 113-138. doi: 10.1177/01436244211020465.
[27] SNK 2.01.18-24. (2025). *Energy consumption standards for heating, ventilation, and air conditioning of buildings and structures*. Retrieved from https://main.tmsiti.uz/media/FILES/shnk/%D0%A8%D0%9D%D2%9A_2.01.18-24.pdf.
[28] SNK 2.08.02-23. (2024). *Approval of urban planning standards and regulations for public buildings and structures*. Retrieved from https://main.tmsiti.uz/media/FILES/shnk/%D0%A8%D0%9D%D2%9A_2.08.02-23.pdf.

Муроділ Мадалієв

Доктор наук з гідравлічної інженерії
Ферганський державний технічний університет
150109, вул. Фергана, 86, м. Фергана, Узбекистан
https://orcid.org/0000-0001-9994-8907

Мірзохід Содіков

Магістр
Ферганський державний технічний університет
150109, вул. Фергана, 86, м. Фергана, Узбекистан
https://orcid.org/0009-0004-0991-6039

Методи моделювання для ефективного розподілу теплової енергії в системах опалення громадських будівель

Анотація. Актуальність дослідження зумовлена наявною проблемою нерівномірного розподілу теплової енергії в однотрубних системах опалення з верхньою розводкою в будівлях, що призводило до перегріву верхніх поверхів і недостатнього обігріву нижніх, негативно впливаючи на тепловий комфорт і енергоефективність приміщень. Метою роботи став аналіз методів покращення мікроклімату приміщень і вирівнювання температури шляхом забезпечення рівномірного розподілу теплоносія в однотрубній системі опалення з верхньою розводкою у громадських будівлях. У дослідженні було використано чисельне моделювання в програмному середовищі COMSOL Multiphysics для оцінювання впливу схеми підключення радіаторів на розподіл тепла в системі опалення п’ятиповерхової громадської будівлі. Досліджено сучасну вертикальну однотрубну систему опалення, сформовано та порівняно базову і модифіковану систему з регулюванням за допомогою клапанів на верхніх поверхах. Розраховано локальні теплові потоки, швидкість руху теплоносія та температуру повітря в приміщеннях. Окремо у роботі було оцінено температурні втрати теплоносія в радіаторах на кожному поверсі. Отримані результати засвідчили суттєву нерівномірність теплопостачання в наявній системі, з перегрівом верхніх поверхів і недостатнім нагріванням нижніх. Після впровадження запропонованого рішення ефективність системи зросла на 21,62 %, а температура теплоносія на вході до першого поверху підвищилася з 47,813 °C до 52,594 °C. Практична цінність дослідження полягає в обґрунтуванні ефективного підходу до покращення теплового розподілу в багатоповерхових будівлях шляхом регулювання потоку за допомогою байпасної лінії в однотрубній системі опалення, що сприяє підвищенню теплового комфорту та зниженню енергоспоживання

Ключові слова: теплопередача; CFD-моделювання; радіаторна система опалення; енергоефективність; тепловий комфорт; байпасне регулювання